

Advanced Dynamic Analysis using MSC Nastran

NAS102B Course Notes

September 2013



Legal Information

MSC.Software Corporation reserves the right to make changes in specifications and other information contained in this document without prior notice. The concepts, methods, and examples presented in this text are for illustrative and educational purposes only, and are not intended to be exhaustive or to apply to any particular engineering problem or design. MSC.Software Corporation assumes no liability or responsibility to any person or company for direct or indirect damages resulting from the use of any information contained herein.

Copyright © 2013 MSC.Software Corporation. All Rights Reserved. This notice shall be marked on any reproduction of this documentation, in whole or in part. Any reproduction or distribution of this document, in whole or in part, without the prior written consent of MSC.Software Corporation is prohibited.

The MSC.Software corporate logo, Adams, Dytran, Easy5, Fatigue, Laminate Modeler, Marc, Mentat, MSC Nastran, Patran, MSC, MSC Nastran, Mvision, Patran, SiMSCesigner, SimEnterprise, SimManager, SimXpert and Sofy are trademarks or registered trademarks of the MSC.Software Corporation in the United States and/or other countries. NASTRAN is a registered trademark of NASA. All other trademarks belong to their respective owners.

Contents

Section	Page
1 Overview	
Course Objectives	1-3
Where to go for Help	1-4
Where to get Training	1-8
2 Review of Basic Dynamics Topics	
Nastran Set Definitions	2-4
Mass Properties	2-6
Damping	2-13
Transient Response	2-33
Frequency Response	2-59
OUTPUT(XYPLOT) and PLOTPS	2-69
3 Dynamic Analysis Checkout Tools	
Miscellaneous Model Checkout Tools	3-4
Modal Effective Mass	3-27
Using Strain Energy to Identify Areas for Change	3-38
Grid Point Kinetic Energy	3-43
Weight Check	3-49
Mode Selection	3-56
Modal Contribution Factor	3-61
4 Direct Matrix Input	

Contents

Section		Page
5	Residual Vectors	
	Concept of Modal Approach	5-3
	Residual Vector Method	5-7
	Case Study	5-14
6	Complex Eigenvalue Analysis	
	Complex Eigensolution Theory	6-5
	Complex Eigenvalue Method	6-8
	Example	6-16
	Recommendations	6-21
7	Extra Points, Transfer Functions and NOLINS	
	Extra Points	7-3
	Transfer Functions	7-4
	NOLINS	7-5
8	Test-Analysis Correlation	
	Pre-Test Planning	8-5
	Post-Test Evaluation	8-9
	Model Refinement	8-36

Contents

Section		Page
9	FRF/FBA/TPA	
	FRF/FBA/TPA Concept	9-7
	Implementation	9-16
	Examples	9-56
10	Introduction to Nonlinear Dynamics	
11	Additional Special Topics in Dynamics	
12	Superelements	
13	Rotor Dynamics	
14	Acoustics	
15	Design Optimization	
16	Monitor Points	
17	Adams MNF	

SECTION 1

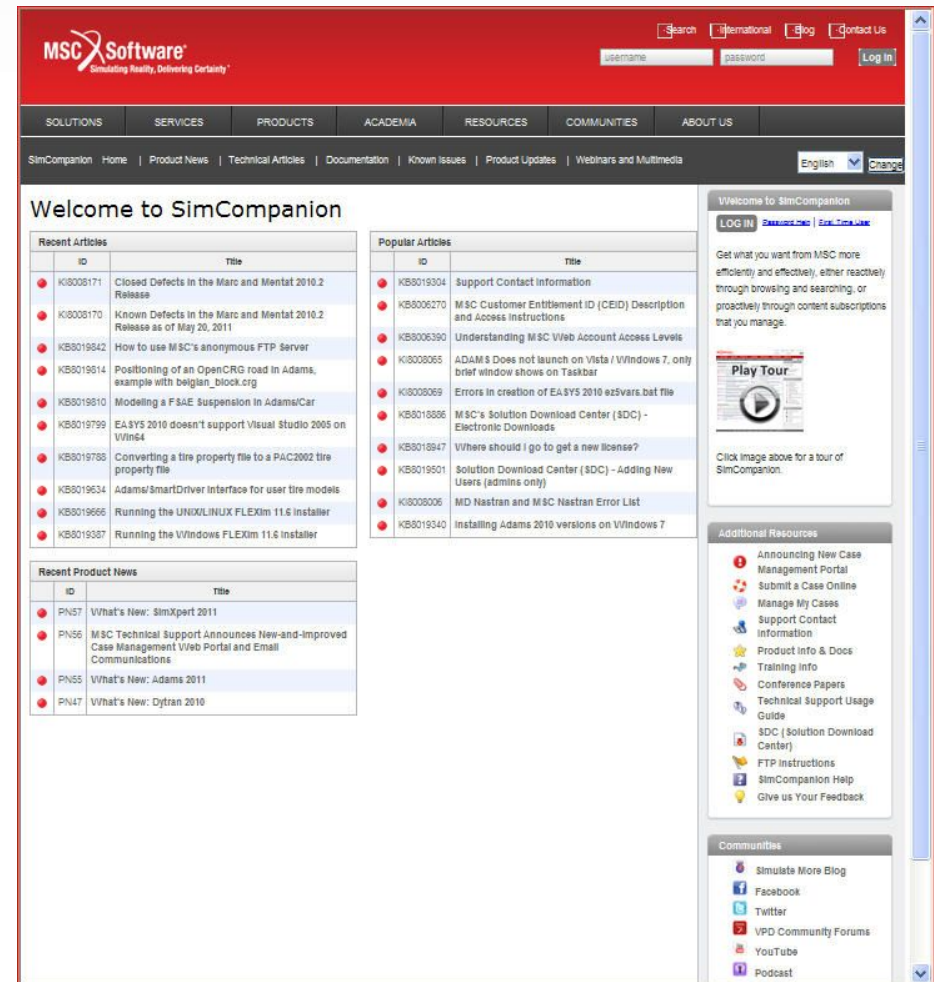
OVERVIEW

Course Objectives

- **Review and expand upon the basic dynamic analysis topics**
- **Introduce model checkout tools for dynamics**
- **Introduce advanced dynamic analysis capabilities**
 - Realize that many of these topics have a complete class of their own, so this is to be considered an introduction
- **Show where to get technical support**
- **Show the available training for MSC Nastran**

SimCompanion

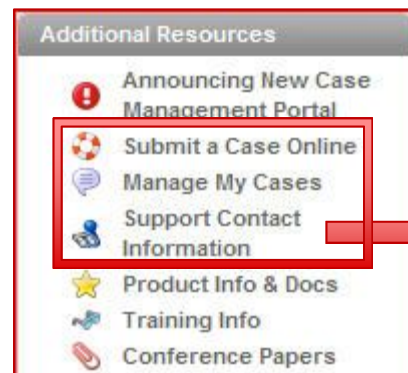
- One stop for full online support
- Find answers to your questions
- Search across ALL content
- Subscribe to email notification
- Single sign-on to ALL content
- Access to other support resources
 - Case Management Portal
 - Discussion Forums
 - Training Information



<http://simcompanion.mscsoftware.com>

SimCompanion (Cont.)

- **Personalized Support via the following channels**
 - Web
 - Submit a Case Online
 - Manage My Cases
 - Email
 - List of Addresses in Support Contact Information
 - Phone
 - List of Phone Numbers in Support Contact Information



Web:	
http://support.mscsoftware.com/servicerequest	
Email:	
English	
- MD Nastran or MSC Nastran	mscnastran.support@mscsoftware.com
- MD Adams or Adams	mscadams.support@mscsoftware.com
- Patran	mscpatran.support@mscsoftware.com
- Dytran	mscdytran.support@mscsoftware.com
- Easy5	easy5.support@mscsoftware.com
- Fatigue	mscfatigue.support@mscsoftware.com
- Marc	mscmarc.support@mscsoftware.com
- Mvision	mscmvision.support@mscsoftware.com
- Sinda	mscsinda.support@mscsoftware.com
- Sofy	mscssofy.support@mscsoftware.com
- SimDesigner	simdesigner.support@mscsoftware.com
- MSC SimManager	simmanager.support@mscsoftware.com
- MSC SimXpert	simxpert.support@mscsoftware.com
Chinese (Simplified)	support.cn@mscsoftware.com
Chinese (Traditional)	support.tw@mscsoftware.com
Dutch	support.be@mscsoftware.com
French	support.fr@mscsoftware.com
German	support.de@mscsoftware.com
Italian	support.it@mscsoftware.com
Japanese	support.jp@mscsoftware.com
Korean	support.kr@mscsoftware.com
Nordic	support.no@mscsoftware.com
Portuguese	suporte.mscbrasil@mscsoftware.com
Russian	support.ru@mscsoftware.com
Spanish	support.es@mscsoftware.com
Phone:	
Belgium and Luxembourg	+31 182 536444
Brazil	0800-891-4346
China - Beijing	+86-10-82607000
China - Shanghai	+86-21-63326655
China - Chengdu	+86-28-86199275/6
China - Shenzhen	+86-755-23811895
Czech Republic	+420 54517 6106
DA/CH	+49 89 431 987 277
Denmark	+45 61 22 32 00
Finland	0800-9-14709
France	+33 5 34 60 44 80

SimCompanion (Cont.)

- **Product Info and Docs**
 - Access to all Product Documentation

Additional Resources

- Announcing New Case Management Portal
- Submit a Case Online
- Manage My Cases
- Support Contact Information
- Product Info & Docs**
- Training Info
- Conference Papers
- Technical Support Usage Guide
- SDC (Solution Download Center)
- FTP Instructions
- SimCompanion Help
- Give us Your Feedback

Documentation

Product Information and Documentation

Back to all Documentation Email Page Printer Friendly Post a comment

Documentation ID: DOC9275
Status: Published
Published date: 09/25/2009
Updated: 01/14/2011

Description
simcompanion.mscsoftware.com/sdc/center/index Click on each link below will take you to a page that provides a summary of Product Information and Documentation for current and prior versions of MSC software products, such as:

- What's New:
 - Release Guides
 - Hardware & Software Requirements
 - Set Up Guides (Installation, Licensing, and Configuration)
 - Other product-specific content.

CAE Tools

Patran MSC Nastran Adams Marc & Mentat MD Adams MD Nastran MSC SimSport™ MSC SimDesigner™ MSC FEA

Additional Resources

- Submit a Request Online
- My Requests
- Support Contact Information
- Product Info & Docs
- Documentation Publishing Group
- Conference Papers
- Technical Support Usage Guide
- SDC (Solution Download Center)
- FTP Instructions
- SimCompanion Help
- Give us Your Feedback

Communities

- Simulate More Blog
- Facebook

Documentation

MD Nastran Product Information & Documentation

Back to all Documentation Email Page Printer Friendly Post a comment

Documentation ID: DOC9279
Status: Published
Published date: 07/21/2010
Updated: 05/11/2011
Reported to: MSC / MD Nastran - MSC / MD Nastran Docs

Description
MD Nastran Product Information & Documentation

	MD Nastran 2011	MD Nastran 2010	MD Nastran R3	MD Nastran R2.1	MD Nastran R2	MD Nastran R1
What's New		What's New: MD Nastran 2010	UNIX & Windows NT	UNIX & Windows NT	UNIX & Windows NT	UNIX & Windows NT
Release Guide	DOC9843	DOC9485	UNIX & Windows NT	UNIX & Windows NT	DOC9519	UNIX & Windows NT
Hardware & Software Requirements	DOC9844 Chap. 1 pg. 5	DOC9486 Chap. 1 pg. 4	App. D	UNIX & Windows NT	UNIX & Windows NT	UNIX & Windows NT
Set Up Guides (Installation, Licensing, & Configuration)	DOC9844	DOC9486	DOC9107	UNIX & Windows NT	UNIX & Windows NT	UNIX & Windows NT
User's Guides						
Getting Started with MD Nastran		DOC9470				
Linear Static Analysis	DOC9846	DOC9469				
Dynamic Analysis	DOC9847	DOC9468				
Quick Reference Guide	DOC9845	DOC9467				
MD Demonstration Problems	DOC9848	DOC9471				
Design Sensitivity and Optimization		DOC9472				
Explicit Nonlinear		DOC9473				
User Defined Services	DOC9850	DOC9474				
EFEA		DOC9475				
EFEA Tutorial		DOC9476				

Additional Resources

- Announcing New Case Management Portal
- Submit a Case Online
- Manage My Cases
- Support Contact Information
- Product Info & Docs
- Training Info
- Conference Papers
- Technical Support Usage Guide
- SDC (Solution Download Center)
- FTP Instructions
- SimCompanion Help
- Give us Your Feedback

Communities

- Simulate More Blog
- Facebook
- Twitter
- VPD Community Forums
- YouTube
- Podcast

SimCompanion (Cont.)

- Access to Communities
 - VPD Community Discussion Forums
 - Subscribe to discussion communities of interest



MSC Software

International Search Sign Out My Account Cart

Simulate More

Products Industry Solutions Services Academic Training & Support Partners Corporate Store

Search Simcompanion My Community Home | FAQ | Getting Started

SimDesigner | SimManager | SimXpert Click here to go to Internal Forums ...

Adams | Dytran | Easy5 | Fatigue | Marc | Nastran | Patran | SimOffice | Sofy | Superforge/Superform | Universities & Student Edition | Private

Main Index | Search | My Profile | My Subscriptions | Who's Online | User List

Webinars	Threads	Posts	Last post
Patran Webinars This forums contains webinar details for MSC Software's Patran product line.	0	0	- New forum

General	Threads	Posts	Last post
Patran 2011 Beta Discussions regarding the testing of the Patran 2011 Beta release.	3 (3 new)	4 (4 new)	Re: WildFire 5 (Mahmud_javadi) - 05/31/11 04:33 PM
Interface, CAD, and Geometry Creation Discussions related to the Patran interface, importing of CAD geometry such as ACIS, PARASOLID, and IGES; creation of patran geometry and manipulation of CAD geometry.	373 (6 new)	1149 (7 new)	Patran 2011 External Beta ... (Mahmud_javadi) - 05/23/11 11:00 AM
Elements, Loads, and Boundary Conditions Discussions regarding meshing techniques, element creations (including MPCs, Superelements, ASETs, QSET, etc.), loads and constraints applications.	560 (10 new)	1624 (29 new)	Re: Meshing big, complex S... (lasmorten) - 05/26/11 11:49 AM
Materials, Properties, and Fields Discussions related to materials, properties (including composites), and fields.	175 (2 new)	467 (2 new)	Effective in-plane engine... (gelievable) - 03/17/11 01:03 AM
Post Processing Discussions dealing with results processing.	261 (168 new)	703 (444 new)	Re: Reading element stress... (Leedom) - 03/31/11 07:27 PM
Flightloads Discussions dealing with the Flightloads aeroelastic graphical user interface. For discussions focussed on the computational aspects of aeroelasticity, please participate in the MSC Nastran Aeroelasticity forum.	14 (14 new)	46 (46 new)	Re: msc flight loads (shiuvkuderu) - 03/15/11 04:56 PM
Nastran Interface Discussions dealing with the creation and import of Nastran files.	99 (4 new)	262 (5 new)	Re: Modal analysis with ac... (Eagle) - 04/19/11 04:50 PM

MSC Nastran Training

NAS101A	Linear Statics and Normal Modes
NAS101B	Advanced Linear Analysis
NAS102A	Dynamic Analysis
NAS102B	Advanced Dynamic Analysis
NAS104	Nastran Thermal Analysis (SOL 153, 159)
NAS106A	Basic Superelement Analysis
NAS106B	Advanced Superelement Analysis
NAS107	Design Sensitivity and Optimization in Nastran
NAS110	DMAP and Database Applications in Nastran
NAS111	Nastran Aeroelastic Analysis
NAS113	Analysis of Composite Materials with Nastran
NAS115	Fluid-Structure Analysis in Nastran (Acoustics)
NAS120	Linear Static and Normal Modes Analysis Using Nastran and Patran
NAS122	Dynamic Analysis Using Patran and Nastran
NAS123	Nastran Implicit Nonlinear (SOL600) Analysis
NAS124	Thermal Analysis using MSC Nastran (SOL 400)
NAS126	Explicit Nonlinear Analysis (SOL 700)
NAS127	Rotordynamics
NAS133	Linear Contact with MSC Nastran
NAS400	Nonlinear Structural Analysis with MSC Nastran

SECTION 2

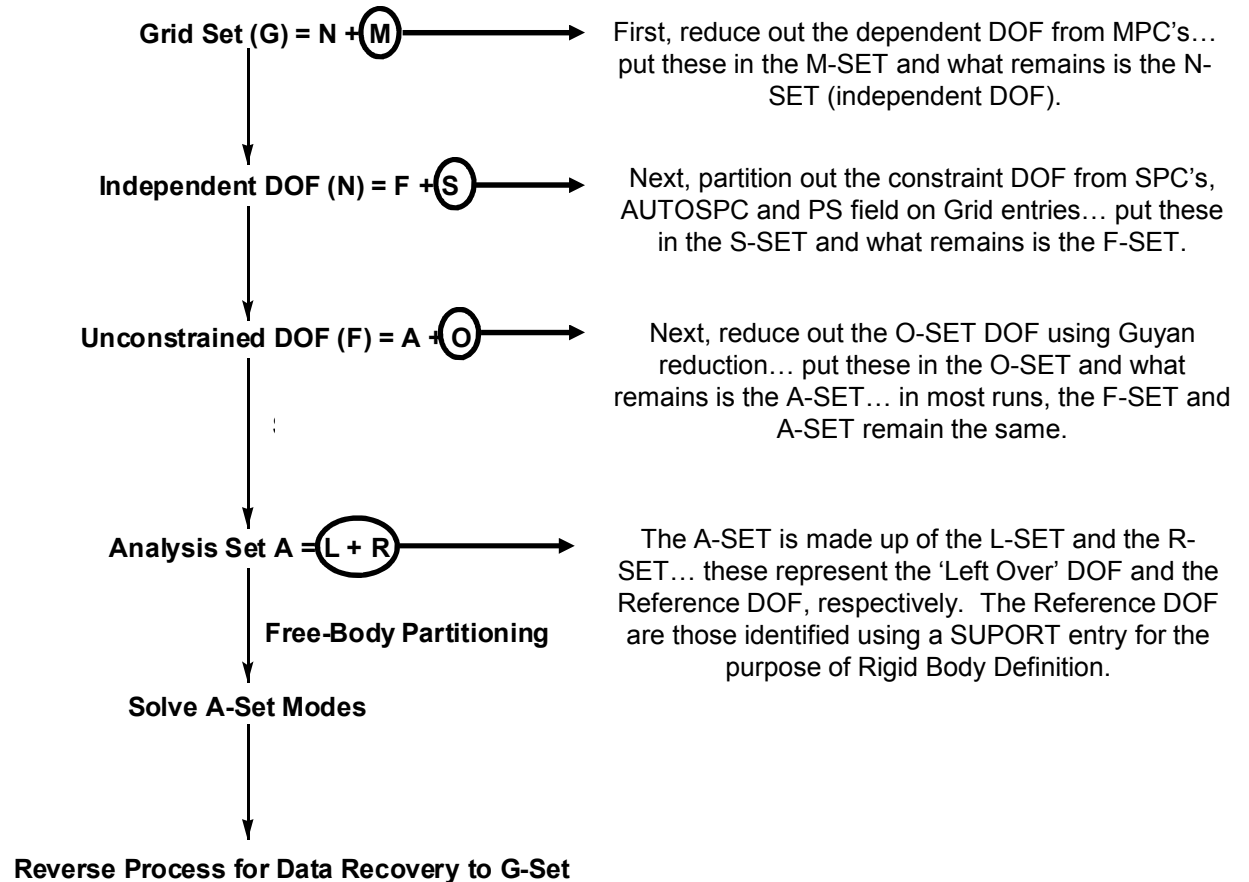
Review of Basic Dynamics Topics

Items to Review

- **MSC Nastran Set Definitions**
- **Mass properties**
- **Damping**
- **Transient Response**
- **Frequency Response**
- **OUTPUT(XY PLOT)**
- **PLOTPS**

BASIC MSC NASTRAN SET OPERATIONS

- See the *MSC Nastran Quick Reference Guide*.



BASIC MSC NASTRAN SET OPERATIONS

- **The set definition defined in the previous page is probably what you will encounter in most situations**
- **See the Degrees-of-Freedom Set Definition section of the Quick Reference Guide for a complete set definition**

Mass Properties

- **When it comes to dynamics, one of the most common errors is improper definition of mass properties**
 - A nice way to visualize the mass distribution in a model is to apply 1-g gravity loading and request the applied loads as data recovery (OLOAD), then use a post-processor to display these loads... $F=Ma$, so the results will be the weight as seen by each grid.
- **Coupled mass is generally more accurate than lumped mass (however, it really needs both methods to be assessed in each case)**
 - As the mesh is refined, the answers should converge between the 2 methods
- **Lumped mass is preferred for computational speed in dynamic analysis**
- **User-selectable coupled mass matrix for elements**
 - PARAM, COUPMASS, 1 to select coupled mass
 - The default is lumped mass

Mass Properties

- **Lumped mass contains only diagonal, translational components (no rotational ones)**
- **Coupled mass contains off-diagonal translational components as well as rotations for BAR (though no torsion for BAR, by default), BEAM, and BEND elements**
 - Setting system cell 398 to 1 generates torsional mass for BAR
 - Setting system cell 398 to 1 and param,coupmass,1 generates consistent axial mass for BAR

NASTRAN SYSTEM(398) = 1 or NASTRAN BARMASS = 1
- **Neglected inertia may result in massless mechanisms**

Mass Properties - Units

- **MSC Nastran assumes you are providing a consistent set of units.**
- **Weight units may be input instead of mass units if this is more convenient. You must convert them using PARAM,WTMASS.**
- **Weight-to-mass conversion:**
Mass = (1/G) Weight (G = Gravity Acceleration)
Mass Density = (1/G) Weight Density
- **PARAM,WTMASS, factor performs conversion with factor = 1/G. The default value for factor is 1.0.**
- **Example:**
 - Input RHO = 0.3 for steel weight density.
 - Use PARAM, WTMASS, 0.00259 (for G = 386.4 in/sec²).
- **PARAM,WTMASS is used once per run and multiplies all weight/mass input (including MASSi, CONMi, and nonstructural mass input). Use all mass or all weight inputs.**

Mass Properties - Input

- **Material density**

- MATi entries

1	2	3	4	5	6	7	8	9	10
MAT1	MID	E	G	NU	RHO	A	TREF	GE	
MAT1	2	30.0E6		0.3	7.76E-4				

- **Scalar mass**

- CMASSi, PMASS

- **Grid point mass**

- CONM1 (6x6 mass matrix) - The user defines half of the terms, symmetry is assumed.
- CONM2 (concentrated mass)

$$\begin{bmatrix} M & & & & & \\ & M & & & & \\ & & M & & & \\ & & & I11 & & \\ & & & -I21 & I22 & \\ & & & -I31 & -I32 & I33 \end{bmatrix} \text{ SYM.}$$

Mass Properties – Input

- **Structural mass (e.g., CONM2, etc.) are always included in the model**
- **Nonstructural mass**
 - Mass input on element property entry which is not associated with geometric properties of element. Input as mass/length for line elements and mass/area for elements with 2-D geometry.
 - Non structural mass can be specified on many property entries (e.g., NSM on the PSHELL entry)
 - This type of nonstructural mass is always included in the model
 - Some nonstructural mass are Case Control selectable
 - NSM Case Control selects NSM, NSM1, NSML, NSML1 Bulk Data entries

Mass Properties – Input, Nonstructural Mass

- **Five nonstructural mass Bulk Data entries—NSM, NSM1, NSML, NSML1, and NSMADD—are available**
 - Distribute nonstructural mass by element lists or specific property lists associated with property entries.
 - Case Control selectable with NSM Case Control command
 - NSM cannot vary by subcase
- **NSM callout can be different between superelements—but only one per superelement**

Mass Properties – Input, Nonstructural Mass

- The NSML and NSML1 entries compute a nonstructural mass coefficient value for “area” elements (e.g., CQUAD4)

$$NSM\_value = \frac{Lumped\_non\_structural\_mass\_value}{\Sigma_{elements} AREA}$$

- The NSML and NSML1 entries compute a nonstructural mass coefficient value for “line” elements (e.g., CBAR)

$$NSM\_value = \frac{Lumped\_non\_structural\_mass\_value}{\Sigma_{elements} LENGTH}$$

- NSML and NSML1 cannot mix “area” and “line” element on the same entry
- NSML and NSML1 are then converted to NSM and NSM1

Damping

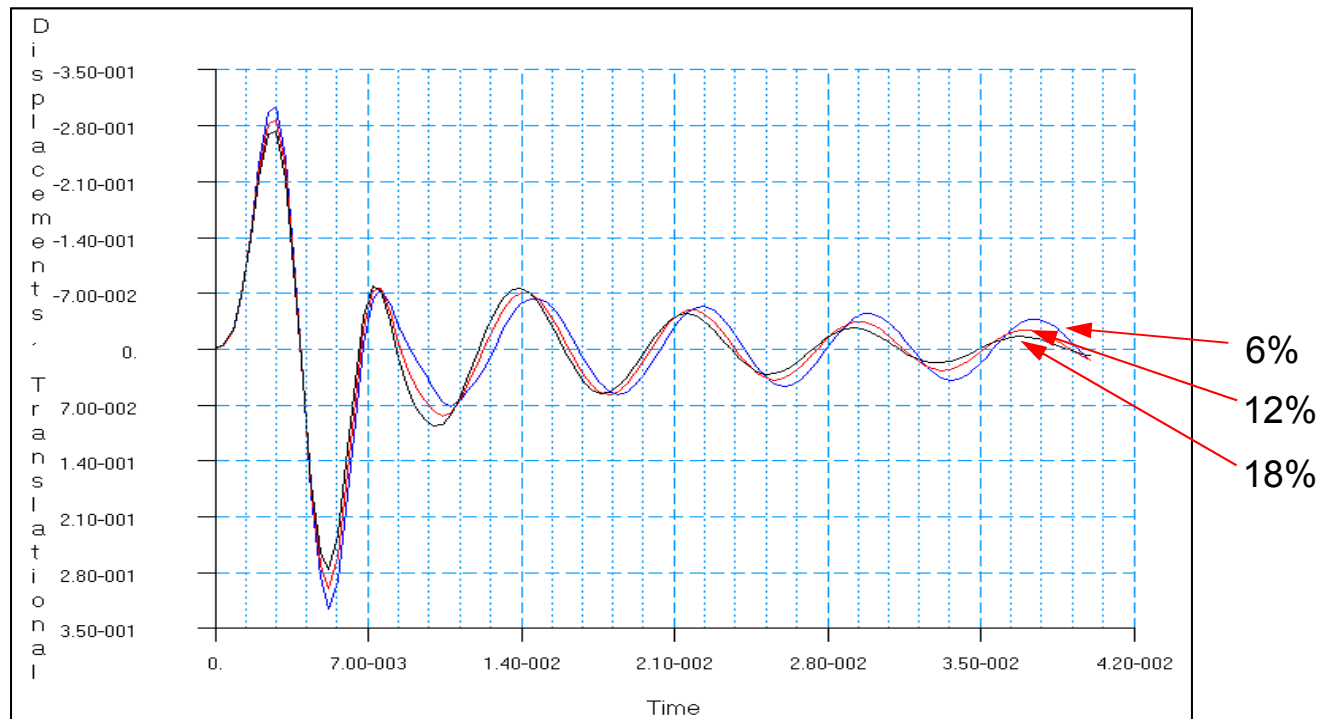
- **Damping is present in all oscillatory systems**
- **Damping represents energy dissipation observed in structures, such that the amplitude and duration of response of the system is reduced**
 - Mechanical motion, Heat, Sound waves, Fluid motion
- **Resulting effect**
 - Free Vibration gives decay in amplitude
 - Steady State gives energy loss = energy input
- **Damping is difficult to accurately model since damping results from many mechanisms:**
 - Viscous effects (dashpot, shock absorber)
 - External friction (slippage in structural joints)
 - Internal friction (characteristic of material type)
 - Structural nonlinearities (plasticity)

Damping

- **What levels of damping should be used? There is no one good answer to this question!**
- **For a particular structural configuration and material look at:**
 - Test results; Industry standards; Company experience
- **Remember, in general, lower levels of damping are more conservative**
 - Use a range of damping values in presenting results
 - Peak response is difficult to capture
- **Typical values (NOT TO BE QUOTED !)**
 - NC machined components : 2% to 5% of critical
 - Fabricated metal components : 4% to 10% of critical
 - Composites : 6% to 20% of critical

Damping

- If the amount of damping is varied, the variation is visible in the response



Damping

- **Checking Damping Results**

- The damping present in a transient analysis should be checked using the logarithmic decrement method for small damping
- $G=d/\pi$ where, $d = \ln (x1/x2)$ $x1$ and $x2$ are successive amplitudes

- **In the previous slide, the values are**

- $G1=.056$ (blue curve)
- $G2=.116$ (red curve)
- $G3=.174$ (black curve)

Which match very well with the values input

Viscous Damping Input

- **Scalar viscous damping**

- CDAMPi -- scalar damper ($i = 1,2,3,4,5,6$) between two DOFs, with reference to a property entry (PDAMP) or enter value of B on CDAMPi entry
- CVISC -- element damper between two grid points; references a property entry (PVISC)
- CBUSHi -- generalized spring and damper element that may be defined as nonlinear or frequency dependent. Damping values are assigned via physical property input on a PBUSHi entry.
 - PBUSH
 - Viscous damping
 - Structural damping
 - PBUSH1D
 - Viscous damping
 - Linear or nonlinear properties

Structural Damping Input

- **Structural damping**

- MATi entries

1	2	3	4	5	6	7	8	9	10
MAT1	MID	E	G	NU	RHO	A	TREF	GE	
MAT1	2	30.0E6		0.3				0.10	

- **PARAM,G, factor (Default = 0.0)**
 - Overall structural damping coefficient to multiply entire system stiffness matrix
- **PARAM,W3, factor (Default = 0.0)**
 - Converts overall structural damping to equivalent viscous damping (transient response)
- **PARAM,W4, factor (Default = 0.0)**
 - Converts element structural damping to equivalent viscous damping (for transient response)
- **Units for W3,W4 are radians/unit time**
- **If PARAM,G or GE are used, PARAM,W3/W4 must be given a value greater than zero; otherwise, PARAM,G and GE are ignored**

Structural Damping versus Viscous Damping

- Assume sinusoidal response: $u = \bar{u}e^{i\omega t}$

$$\dot{u} = i\omega\bar{u}e^{i\omega t}$$

$$\ddot{u} = -\omega^2\bar{u}e^{i\omega t}$$

Then:

- Viscous damping:

$$m\ddot{u} + b\dot{u} + ku = p(t)$$

$$m(-\omega^2\bar{u}e^{i\omega t}) + b(i\omega\bar{u}e^{i\omega t}) + k\bar{u}e^{i\omega t} = p(t)$$

$$-\omega^2 m\bar{u}e^{i\omega t} + \underbrace{ib\omega\bar{u}e^{i\omega t}} + k\bar{u}e^{i\omega t} = p(t)$$

- Structural damping:

$$m\ddot{u} + (1 + ig)ku = p(t)$$

$$m(-\omega^2\bar{u}e^{i\omega t}) + (1 + ig)k\bar{u}e^{i\omega t} = p(t)$$

$$-\omega^2 m\bar{u}e^{i\omega t} + \underbrace{igk\bar{u}e^{i\omega t}} + k\bar{u}e^{i\omega t} = p(t)$$

Structural Damping versus Viscous Damping

- Both equations are identical if:

$$gk = b\omega \rightarrow b = \frac{gk}{\omega}$$

Therefore, if structural damping g is to be modeled using viscous damping b , then the equality holds at only one frequency ω_n (or ω_d).

if $b = \frac{gk}{\omega}$

$$\omega = \omega_n = \sqrt{\frac{k}{m}}$$

but $b = \frac{gk}{\omega_n} = g\omega_n m$

$$b_c = 2m\omega_n$$

Structural Damping versus Viscous Damping

then $\frac{b}{b_c} = \zeta = \frac{g}{2}$

ζ = critical damping ratio (percent critical damping)

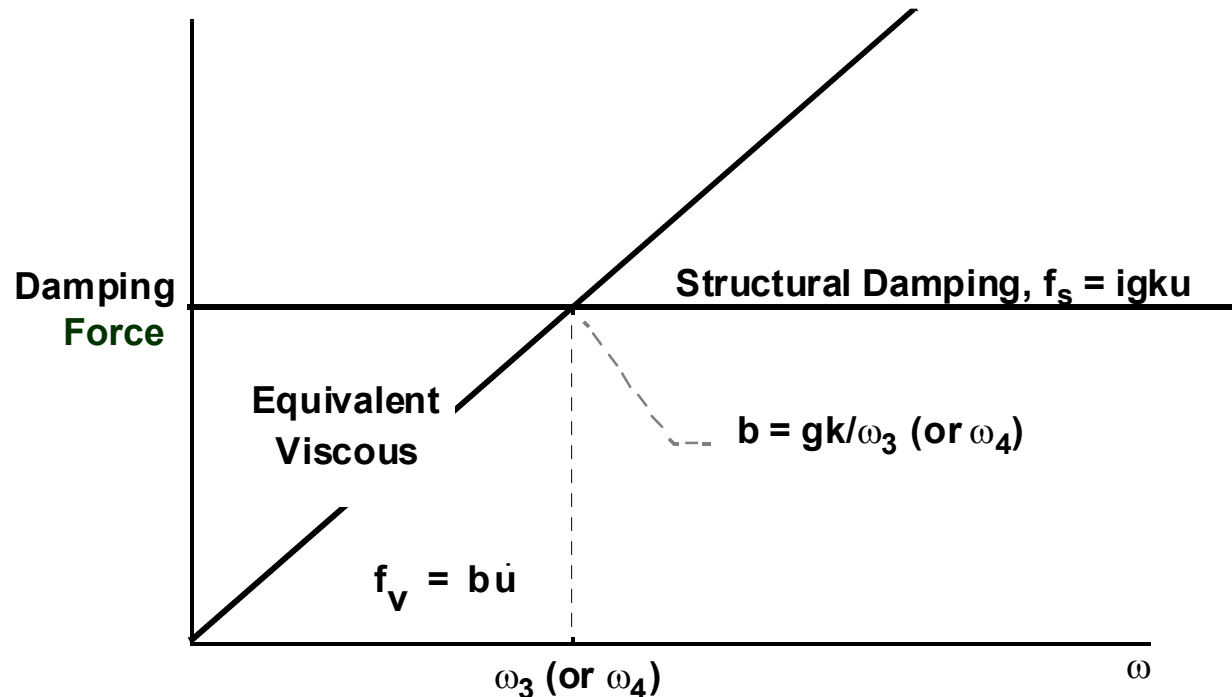
- The smallest amount of damping required to return a system to its equilibrium condition without oscillating.

$g = \frac{1}{Q}$ = structural damping factor

Q = quality factor or magnification factor or amplification factor

- The amount of mechanical gain of a structure when excited at a resonant frequency. The ratio of the amplitude of the steady state solution (amplitude at resonance) to the static deflection for the same force F . This amplification factor is a function of the system damping. For a damping ratio $\zeta = 0$ (no damping) the amplification factor is infinite, for $\zeta = 1$ (critically damped) there is no amplification. Quality or amplification factor can be expressed as $Q = 1/g = 1/2 \zeta$.

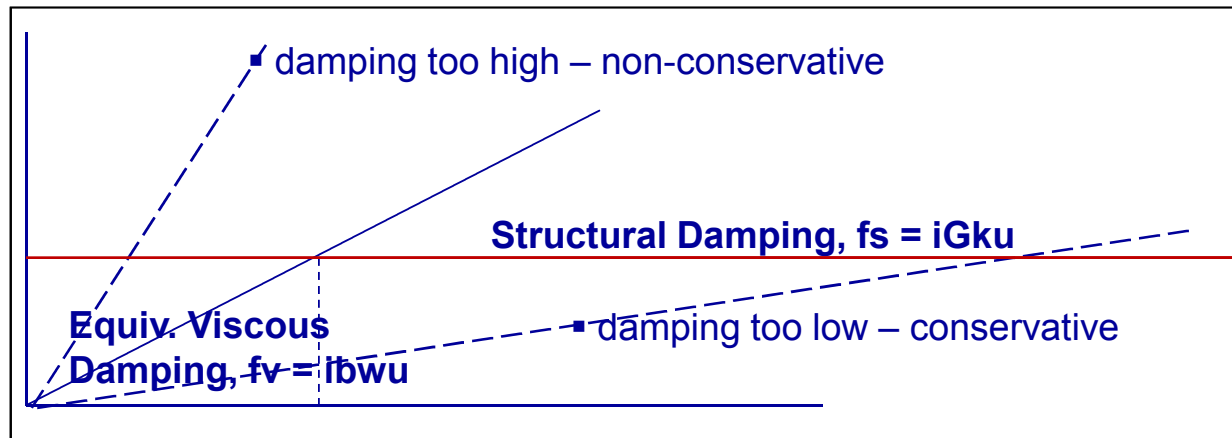
Structural Damping versus Viscous Damping



- For constant displacement, viscous and structural damping are equivalent at frequency ω_3 (or ω_4).

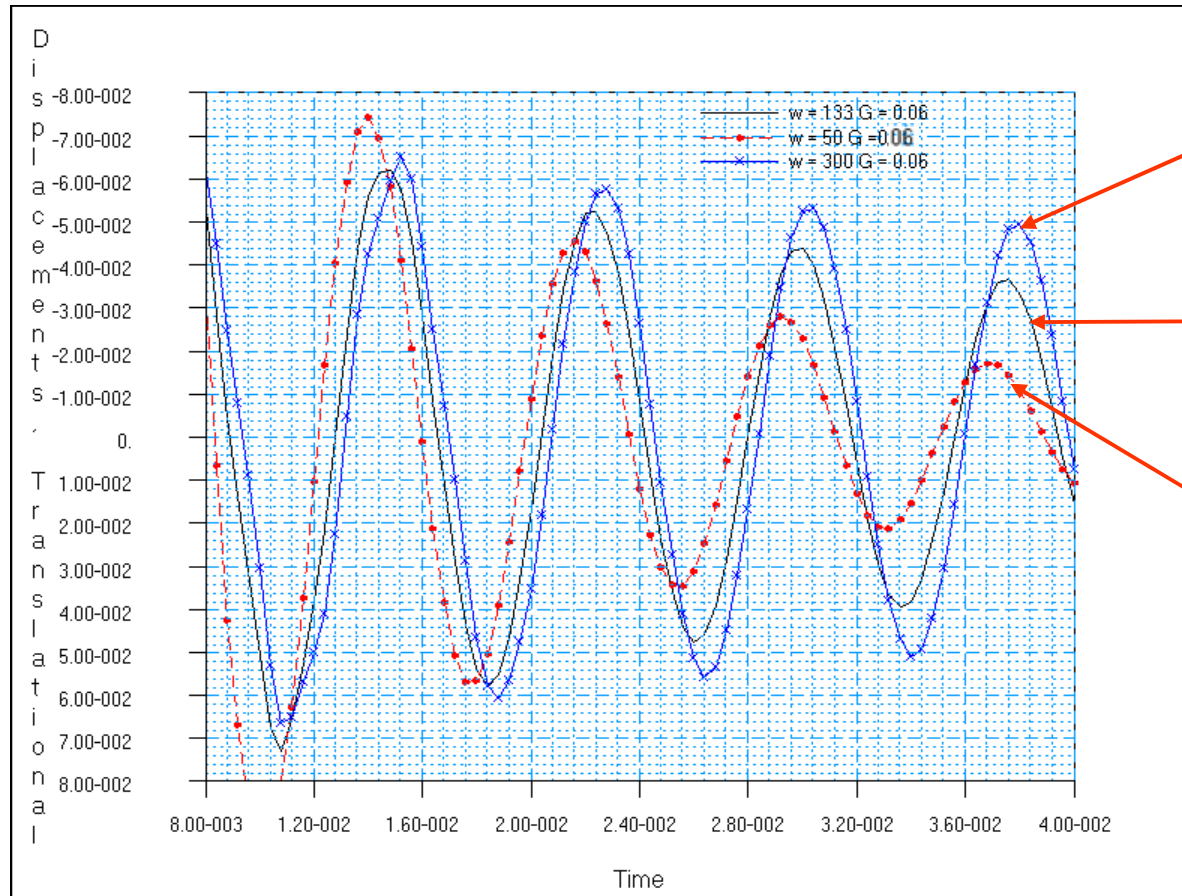
Structural Damping versus Viscous Damping

- **Checking Damping Results**
 - One of the common errors in defining structural damping is to calculate W3 or W4 wrongly (maybe by forgetting to input as rad/sec and incorrectly using Hz instead)
- **The influence of the error can be predicted graphically:**



Structural Damping versus Viscous Damping

- **Transient Damping effect of errors in W3**



W3=300 Hz
Damping is too low

W3=133 Hz

W3=50 Hz
Damping is too high

Modal Damping

- Modal damping can be thought of as ‘catch-all’ damping, since it is associated with a particular mode, not the physical structure
 - Can be obtained from modal tests
- Modal damping is used to represent any source of system damping
 - Depends on the type of excitation, e.g. transient or frequency
 - Example, modal transient analysis with just modal damping; there are no viscous elements, e.g. CVISC, and no structural damping:

$$\ddot{\xi}_k(t) + 2\boxed{\zeta_k}\omega_k\dot{\xi}_k(t) + \omega_k^2\xi_k(t) = N_k(t)$$

where

ζ_k is defined using the TABDMP1 Bulk Data entry

Modal Damping Input

CASE CONTROL

SDAMP = n \$ selects the modal damping table to be used.

BULK DATA

TABDMP1,n,CRIT \$ Lists damping values (in "G", "CRIT", or "Q") versus
,x1,y1,x2,y2,..ENDT \$ frequencies

Rayleigh Damping

- Proportional to either the mass and/or stiffness matrix
- Also known as proportional damping
- Proportional to mass matrix (param,alpha1,x)
- Proportional to stiffness matrix (param,alpha2,y)
- Available in transient and frequency response analysis
- Scale factors applied to d-set (direct) and h-set (modal)
- Added to the viscous damping matrix as follows:

$$[B'] = [B] + \alpha 1 * [M] + \alpha 2 * [K]$$

Rayleigh Damping

- **ALPHA1 and ALPHA2 are Complex parameters**

Real , Imaginary

PARAM, ALPHA2, 1.25E-4, 0.

- **Modal Damping Matrix reads**

$$\begin{aligned}\boldsymbol{\phi}^T \mathbf{B} \boldsymbol{\phi} &= \alpha_1 \boldsymbol{\phi}^T \mathbf{M} \boldsymbol{\phi} + \alpha_2 \boldsymbol{\phi}^T \mathbf{K} \boldsymbol{\phi} \\ &= \alpha_1 \mathbf{I} + \alpha_2 \boldsymbol{\Omega}^2\end{aligned}$$

Rayleigh Damping

- **ALPHA1 and ALPHA2 are real valued parameters, however the programming treats them as complex valued variables, e.g.**

PARAM, ALPHA2, 1.25E-4, 0.0

- **Relationship of modal damping to Rayleigh damping**

- Modal damping

$$\{\ddot{\xi}\} + [2\zeta\omega]\{\dot{\xi}\} + [\omega^2]\{\xi\} = [\phi]^T \{P(t)\}$$

- Rayleigh damping

$$[B] = \alpha_1[M] + \alpha_2[K]$$

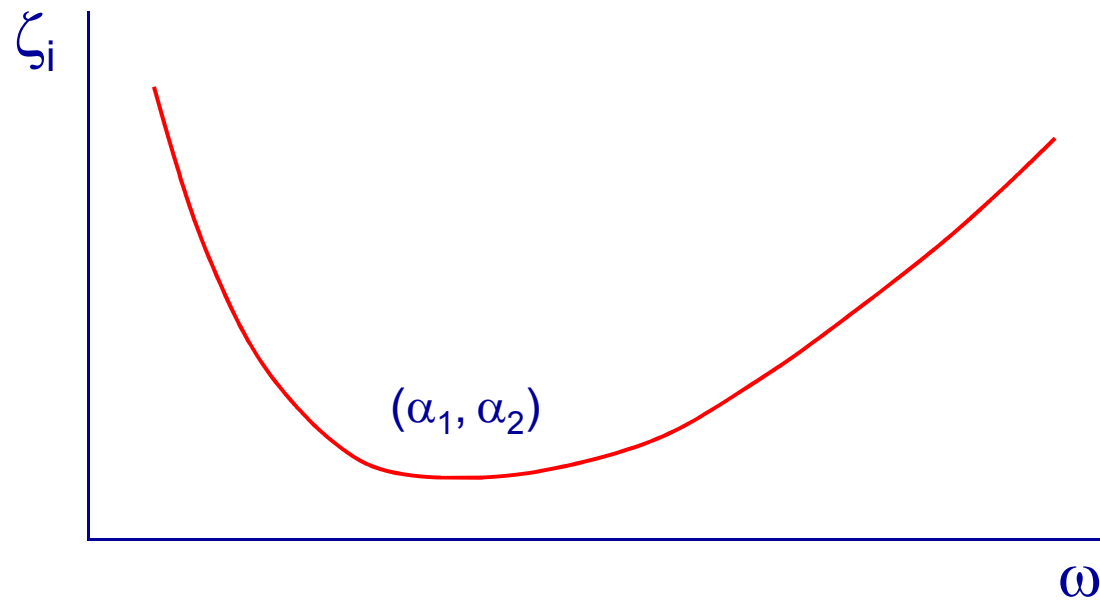
- Solution of the equations is equal for the following damping relationship

$$\{\ddot{\xi}\} + [\phi]^T (\alpha_1[M] + \alpha_2[K])[\phi]\{\dot{\xi}\} + [\omega^2]\{\xi\} = [\phi]^T \{P(t)\}$$
$$[2\zeta\omega] = [\phi]^T (\alpha_1[M] + \alpha_2[K])[\phi]$$

Rayleigh Damping

The relationship between modal and Rayleigh damping is obtained:

$$\zeta_i = \alpha_1 / (2\omega_i) + \alpha_2 \omega_i / 2$$



Damping Summary

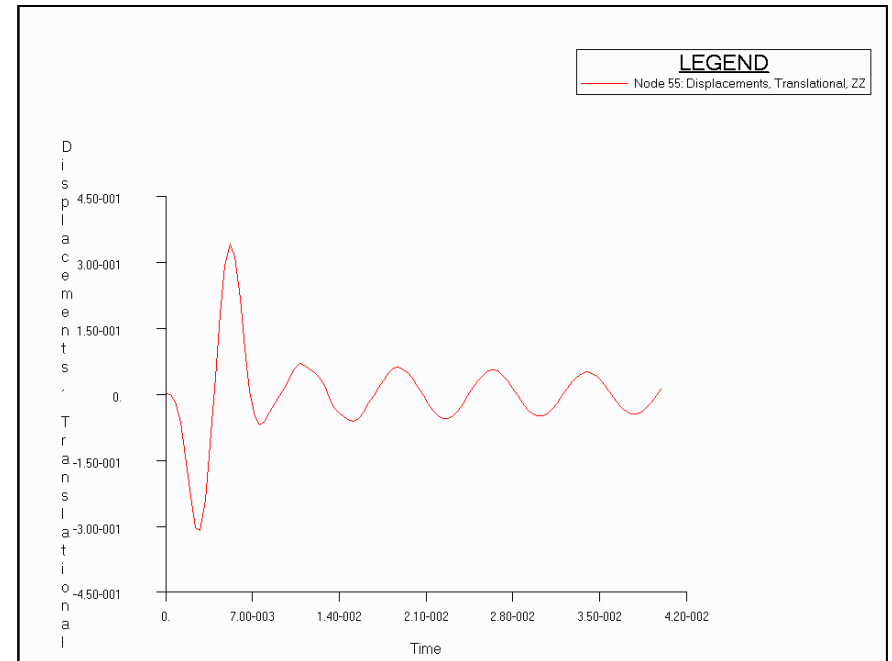
- **Viscous damping force proportional to velocity**
- **Structural damping force proportional to displacement**
- **Modal damping improves efficiency**
- **Critical damping ratio $b/b_{cr} \equiv \zeta$**
- **Quality factor Q inversely proportional to energy dissipated per cycle of vibration**
- **At resonance $\omega \cong \omega_n$**
 - $\zeta = g/2$
 - $Q = 1/(2\zeta)$
 - $Q = 1/g$

Damping Summary

- If no structural damping or viscous damping exists (param G, GE, CDAMP/CVISC etc.) then the problem is reduced to modal coordinates and the modal damping table supplies diagonal damping for each frequency. This allows the model to be solved using uncoupled equations in modal coordinates.
- If the model has structural damping or viscous damping elements, the model is reduced to modal coordinates, modal damping is added if present, and the reduced matrices are solved with the direct solver since the damping matrix is not diagonal (thus coupled solution is required).

Transient Response

- Compute a response to a time-varying input $\{P(t)\}$
- The excitation is explicitly defined in the time domain. All of the applied forces are known at each instant in time.
- Computed response usually includes nodal displacements and accelerations, and element forces and stresses.
- All responses are a function of time.
- There are two categories of analysis
 - Direct
 - Modal



Transient Response – Direct Method

- The direct method uses a brute force solution of the equation of motion using the full matrices
- Numerical solution of dynamic equation

$$[A_1]\{u_{n+1}\} = \{A_2\} + [A_3]\{u_n\} + [A_4]\{u_{n-1}\}$$

– where

$$[A_1] = [M / \Delta t^2 + B / (2\Delta t) + K / 3]$$

Dynamic matrix

$$\{A_2\} = (1 / 3)\{P_{n+1} + P_n + P_{n-1}\}$$

Applied force

$$[A_3] = [2M / \Delta t^2 - K / 3]$$

$$[A_4] = [-M / \Delta t^2 + B / (2\Delta t) - K / 3]$$

} For previous
time points

– Write the solution equation as

$$[\tilde{K}]\{u_{n+1}\} = \{\tilde{P}_{n+1}\}$$

- This is similar to the linear static equation.

Transient Response – Direct Method

- **Solve this equation by decomposing $[A_1]$.**
- **This is similar to the classical Newmark-Beta method, except**
 - The internal forces (element forces) are averaged over 3 adjacent time points. This provides a nonsingular dynamic matrix, $[A_1]$, for $[M]$ and $[B]$ equal to zero.
 - The applied force is averaged over 3 adjacent time points.
- **$[M]$, $[B]$, and $[K]$ do not change with time.**
- **$[A_1]$ needs to be decomposed only once if Δt is unchanged throughout the entire solution. If Δt is changed, $[A_1]$ must be re-decomposed, which may be a computationally time consuming operation.**

Transient Response – Modal Method

- **Now, look at an alternative method of carrying out Transient Analysis, using Modal Coordinates**
- **It has been witnessed that for very large models, with many time steps, the cost of a Direct Transient becomes high**
- **One way to reduce the cost is to use the principle of Modal Transformation. If relatively few Modal Coordinates are used to represent the Physical Coordinates, the problem size can be reduced significantly.**

Transient Response – Modal Method

- Transform from modal to physical coordinates $\{u(t)\} = [\phi] \{\xi(t)\}$

- Ignore damping temporarily, $[B] = [0]$

$$[\phi]^T [M] [\phi] \{\ddot{\xi}(t)\} + [\phi]^T [K] [\phi] \{\xi(t)\} = [\phi]^T \{P(t)\}$$

where $[\phi]^T [M] [\phi]$ is the modal mass matrix; it is diagonal
 $[\phi]^T [K] [\phi]$ is the modal stiffness matrix; it is diagonal
 $[\phi]^T \{P(t)\}$ is the modal force vector

- Write the dynamic equation as a decoupled system (can solve easily)

$$\ddot{\xi}_k(t) + \omega_k^2 \xi_k(t) = N_k(t)$$

where:

ξ_k is the k-th modal coordinate

ω_k is the k-th modal frequency

N_k is the k-th modal force

Transient Response – Modal Method

- Transformation from modal space to physical space, as before, but do not ignore damping
- Viscous damping (no structural damping)

$$[M]\{\ddot{u}(t)\} + [B]\{\dot{u}(t)\} + [K]\{u(t)\} = \{P(t)\}$$

$$[\phi]^T [M] [\phi] \{\ddot{\xi}\} + [\phi]^T [B] [\phi] \{\dot{\xi}\} + [\phi]^T [K] [\phi] \{\xi\} = [\phi]^T \{P(t)\}$$

– $[\phi]^T [B] [\phi]$ is diagonal

$$\ddot{\xi}_k(t) + 2\zeta_k \omega_k \dot{\xi}_k(t) + \omega_k^2 \xi_k(t) = N_k(t)$$

- where
 - ξ_k is the k-th modal coordinate
 - ω_k is the k-th modal frequency
 - ζ_k is the k-th modal damping ratio
 - N_k is the k-th modal force

Transient Response – Modal Method

- If $[\phi]^T[B][\phi]$ is not diagonal
 - The modal coupled problem is solved using modal coordinates and utilizing direct transient response Newmark-Beta type numerical integration.

$$[A_1]\{\xi_{n+1}\} = \{A_2\} + [A_3]\{\xi_n\} + [A_4]\{\xi_{n-1}\}$$

Where

$$[A_1] = [\phi]^T [M / \Delta t^2 + B / (2\Delta t) + K / 3] [\phi]$$

$$\{A_2\} = (1 / 3)[\phi]^T \{P_{n+1} + P_n + P_{n-1}\}$$

$$[A_3] = [\phi]^T [2M / \Delta t^2 - K / 3] [\phi]$$

$$[A_4] = [\phi]^T [-M / \Delta t^2 + B / (2\Delta t) - K / 3] [\phi]$$

- There is still an advantage over the Direct Method, since working with smaller matrices

Transient Response – Modal Method, Data Recovery

- Recover physical response, e.g. $\{u\}$, as the summation of the modal response

$$\{u\} = [\phi] \{\xi\}$$

$$\{\dot{u}\} = [\phi] \{\dot{\xi}\}$$

$$\{w\} = [\phi]_w \{u\}$$

– where

$\{w\}$ is a vector of internal forces

$[\phi]_w$ is a transformation matrix

TSTEP Entry

- The TSTEP Bulk Data entry controls solution and output Δt , and is selected by the TSTEP Case Control command.
- Selects integration time step for direct and modal transient response.
 - Integration errors increase with increasing natural frequency.
 - The smaller the value of Δt , the more accurate the integration will be.
 - The time step chosen should be sufficiently small to capture the highest frequency of interest in the *response*. For example, if this value is 100 Hz, each time period is 0.01s (10ms), so at the very least, 5 steps to capture the response are needed, i.e. $\Delta t = 0.002\text{s}$ (2ms). The preferred minimum number is 10 steps per period.
 - The accuracy of the *input load* is similarly dependent on the time step chosen, so a loading of 1000 Hz will need at least a Δt of 0.0002s (0.2ms).

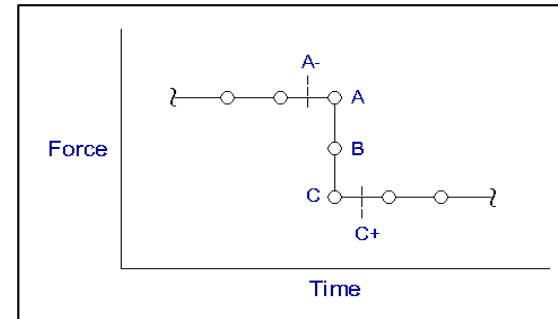
TSTEP Entry

- **The cost of integration is directly proportional to the number of time steps when Δt is constant.**
 - Keep Δt constant if possible. If Δt is changed, $[A_1]$ must be re-decomposed (which may be a time consuming operation).
 - This controls the duration of the analysis. The duration should be sufficient to ensure the response has reached steady state and is decaying in a predictable manner.
 - The lowest significant frequency needs to be checked to ensure enough decay time has been allowed. Assume the lowest frequency is 10 Hz, then its time period is 100ms. The analysis needs to be *extended* to, say, 300ms (300 time steps of 1ms), to ensure the long period motion is captured.
- **If nonlinearities described by NOLIN entries exist, Δt should be 1/5 to 1/10 the size described previously. This is because the NOLINi puts the structure out of equilibrium by one time step, so a very small integration time step is needed to minimize this approximation.**

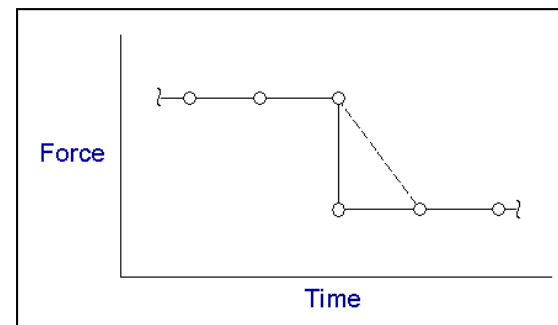
Transient Response – Excitation Issues

- There is a 1/3 “smearing” of applied loads. This will smooth the force, decreasing its apparent frequency content.
- Avoid specifying discontinuous forces. Having this type of force may cause different results on different computers.

- The results from integration will differ depending on whether the force at $N\Delta t$ is at A, at B, or at C.

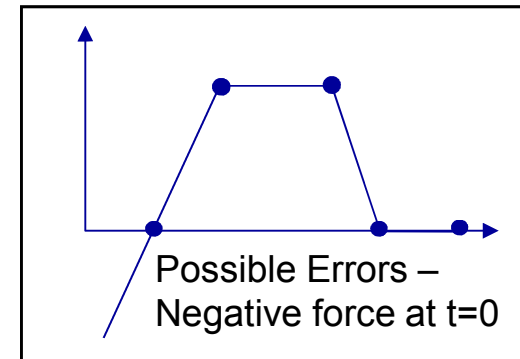
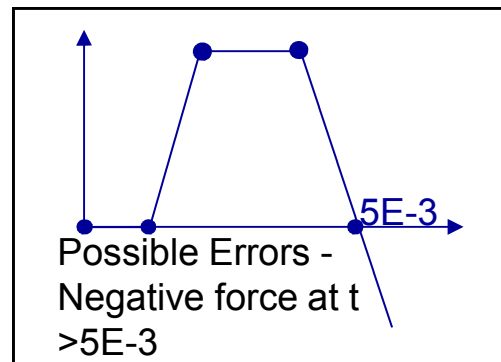
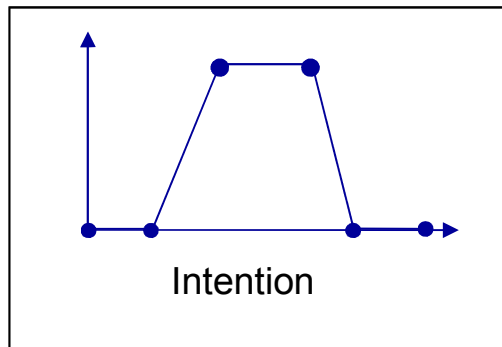


- Smooth a discontinuous force over one Δt .



Transient Response – Excitation Issues

- When defining an input table, ensure that the data extrapolates correctly. The MSC Nastran rule for any TABLE is to extrapolate the first 2 and the last 2 data points. Poorly defined data points can lead to bad extrapolation.



Means of Defining Transient Excitation

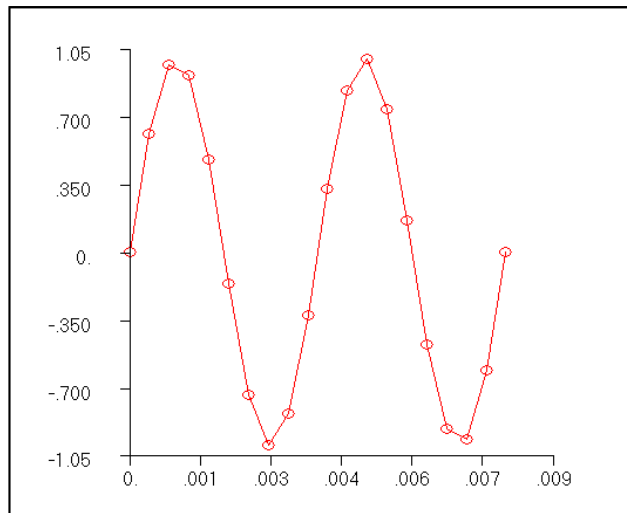
- **Define force as a function of time**
- **Primary methods in MSC Nastran:**
 - TLOAD1 “Brute force”; ordered time, force pairs table input
 - The input time history is entered at discrete time steps with a TABLED1 entry.
 - TLOAD2 Efficient definition for analytical-type loadings
 - Can be used when the input load is in the form of a simple function, such as a cosine function.

TLOAD1

- The TLOAD1 Bulk Data entry is 'activated' by either the Case Control DLOAD entry, or the DLOAD Bulk Data entry
- The TLOAD1 entry in turn points to the 'excite ID' and the TABLED1 defining the time history tabular data
- The 'excite ID' can be any static type load entry (FORCE, PLOAD4, GRAV, etc), as well as a DAREA or SPCD entry
- A flag exists to indicate whether the form of the loading is force, enforced displacement, enforced velocity or enforced acceleration

TLOAD1 Entry

- TLOAD1 SID 5 is linked to TABLED1 TID 1
- The table contains the Time vs. Force pairs which make up the time dependency



TLOAD1	5	6	1

\$ Dynamic Load Table : force				
TABLED1	1			
*	-8.-4	0.	0.	0.
*	4.21053-4	-.61421	8.42105-4	-.9694
*	.00126316	-.91577	.00168421	-.47594
*	.00210526	.164595	.00252632	.735724
*	.00294737	.996584	.00336842	.837166
*	.00378947	.3247	.00421053	-.32469
*	.00463158	-.83716	.00505263	-.99658
*	.00547368	-.73572	.00589474	-.16459
*	.00631579	.475948	.00673684	.915774
*	.0071579	.9694	.00757895	.614212
*	.008	-2.1305-6	.0088	-2.1305-6
*	ENDT			

TLOAD2 Entry

- The TLOAD2 follows exactly the same pattern as the TLOAD1 in linking to other data entries.
- It permits direct entry of sine, cosine, exponential, power series and step type functions and can be useful when the loading specification calls for this type of input.
- Examples of its usage are:
 - A shock acceleration input of form half sine pulse at 10Hz, amplitude 15g applied from 0.0 to 0.05 secs
 - A continuous sinusoidal loading of 250 Hz, amplitude 3 lbf applied from 0 to 40 secs

TLOAD2 Entry

- For defining excitation in the form of

$$\{P(t)\} = \begin{cases} \{0\} & , t < (T1 + \tau) \text{ or } t > (T2 + \tau) \\ \{A\} \tilde{t}^B e^{C\tilde{t}} \cos(2\pi F\tilde{t} + P) & , (T1 + \tau) \leq t \leq (T2 + \tau) \end{cases}$$

where

$$\tilde{t} = t - T1 - \tau$$

- For a constant load, leave fields F, P, C, and B blank.
- For a cosine wave, specify F = 1.0, and leave fields P, C, and B blank.
- For a sine wave, specify F = 1.0, P = - 90° and leave fields C and B blank.

Load Set Combination- DLOAD

- The applied load $\{P_c\}$ is constructed from a combination of component load sets $\{P_k\}$

- where

$$\{P_c\} = S \sum_k S_k \{P_k\}$$
 - S = overall scale factor
 - S_k = scale factor for k-th load set
 - $\{P_k\}$ = load vector for k-th load set, with corresponding SID of TLOAD entry

1	2	3	4	5	6	7	8	9	10
DLOAD	SID	S	S_1	P_1	S_2	P_2	-etc-		

- TLOAD1s and TLOAD2s must have unique SIDs.
- Use the DLOAD entry to combine TLOADs.
- The DLOAD Bulk Data entry is selected by DLOAD Case Control command.
- Put it all together in the example on the next slide

Transient Response - Initial Conditions

- Initial conditions are available in both direct and modal transient using the IC Case Control command:

PHYSICAL
IC (MODAL) = n
STATSUB [,DIFFK]

- IC (physical or modal) points to TIC entry to specify initial conditions
 - Can be specified in physical or modal coordinates
- Residual vectors **must** be included (default) for modal solution
- Static solutions can be used as initial condition in linear transient
- May include differential stiffness effect
- If using STATSUB in a Modal Transient, the physical IC are automatically transformed to modal IC

Transient Response - Initial Conditions

- In order to use the STATSUB and/or the DIFFK option, a static Subcase must be defined
- Then the dynamics Subcase would use the IC command pointing to the static Subcase ID
- An example is below...

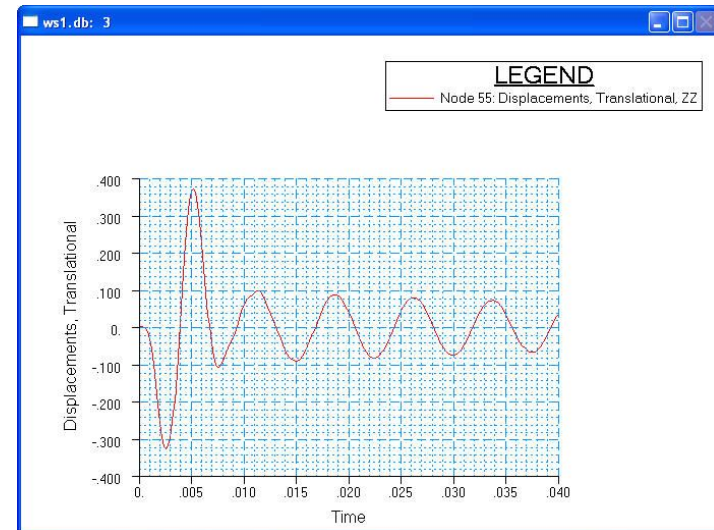
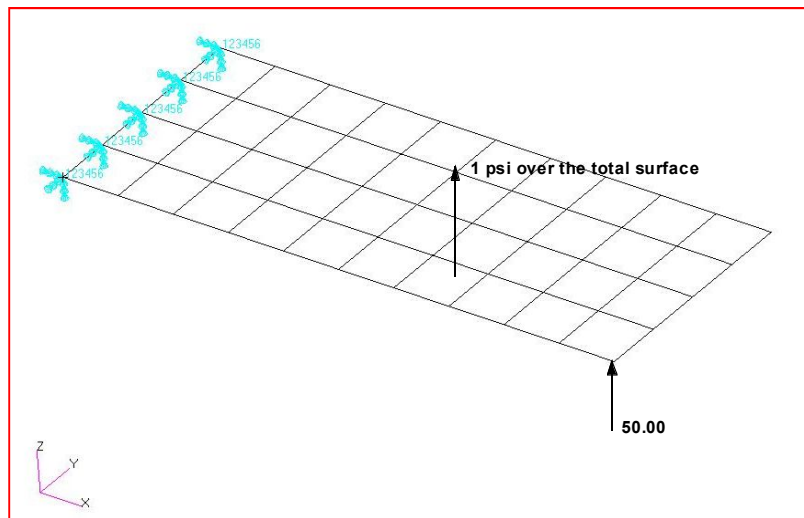
```
SPC=2
SUBCASE 1 $ static subcase
LOAD=5
SUBCASE 2 $ dynamic subcase
DLOAD=10
METHOD=5
IC(STATSUB,DIFFK)=1
```

Transient Response - Restarts

- In MSC Nastran, restarts in linear transient are performed with **“PARAM,STIME,x”**
 - Database from cold start run must be saved (using SCR=NO keyword)
 - No model or BC changes in restart run
 - Content modifications to the DLOAD and TSTEP entries are allowed, but the IDs must remain the same
 - “PARAM,STIME,x” in restart run, where x is the time the run will continue from... can restart from any time step
 - Use the “/” command to delete unwanted items from the cold start run

Transient Response - Restarts

EXAMPLE:



- **Cold Start run – run job to 0.005 sec**
- **Restart job – run job from 0.005 sec to 0.04 sec**

Transient Response - Restarts

Cold Start Run:

Must echo the sorted Bulk Data in order to find and replace the TSTEP entry in the restart run.

Run the analysis to whatever initial time desired...

```
$ Coldstart run to demonstrate the use of STIME for RESTARTs in Linear Transient
$ coldstart.bdf... run to .005 sec
$ Run this file with scr=no keyword option on command line
SOL 109
$ Direct Text Input for Executive Control
CEND
TITLE = MSC Nastran job created on 20-Aug-08 at 15:13:58
ECHO = SORT
LOADSET = 1
SUBCASE 1
  TITLE=This is a default subcase.
  TSTEP = 1
  SPC = 2
  DLOAD = 2
  SET 111 = 55
  DISPLACEMENT(PLOT,SORT2,REAL)=111
  SPCFORCES(PLOT,SORT2,REAL)=ALL
BEGIN BULK
PARAM  POST  0
PARAM  WTMASS .00259
PARAM  G    .06
PARAM  W3   1570.
PARAM  PRTMAXIM YES
$ this delta-t and number of steps will take the analysis to .005 sec
TSTEP  1    50    1.-4
```


Transient Response - Restarts

Restart Run:

Make the file assignment to the previous run (coldstart).

Add the RESTART entry.

Leave the Case Control Section unchanged.

Delete the previous TSTEP entry. Then introduce the new TSTEP.

Add the STIME param indicating the time from which to restart.

\$ Restart run to demonstrate the use of STIME for RESTARTs in Linear Transient

\$ restart.bdf... pick up from .005 sec and continue to .04 sec

ASSIGN RUN1='casestudy_coldstart.MASTER'

RESTART LOGICAL=RUN1

SOL 109

CEND

TITLE = MSC Nastran job created on 20-Aug-08 at 15:13:58

ECHO = NONE

LOADSET = 1

SUBCASE 1

TITLE=This is a default subcase.

TSTEP = 1

SPC = 2

DLOAD = 2

SET 111 = 55

DISPLACEMENT(PLOT, SORT2, REAL)=111

SPCFORCES(PLOT, SORT2, REAL)=ALL

BEGIN BULK

\$ replace the previous TSTEP entry...

/DELETE TSTEP

TSTEP 1 350 1-4

\$ this delta-t and number of steps will take the analysis to .035 sec

\$ the STIME has it starting from .005, so final time will be .04 sec

PARAM, STIME, .005

\$

ENDDATA c76d19b9

Transient Response - Restarts

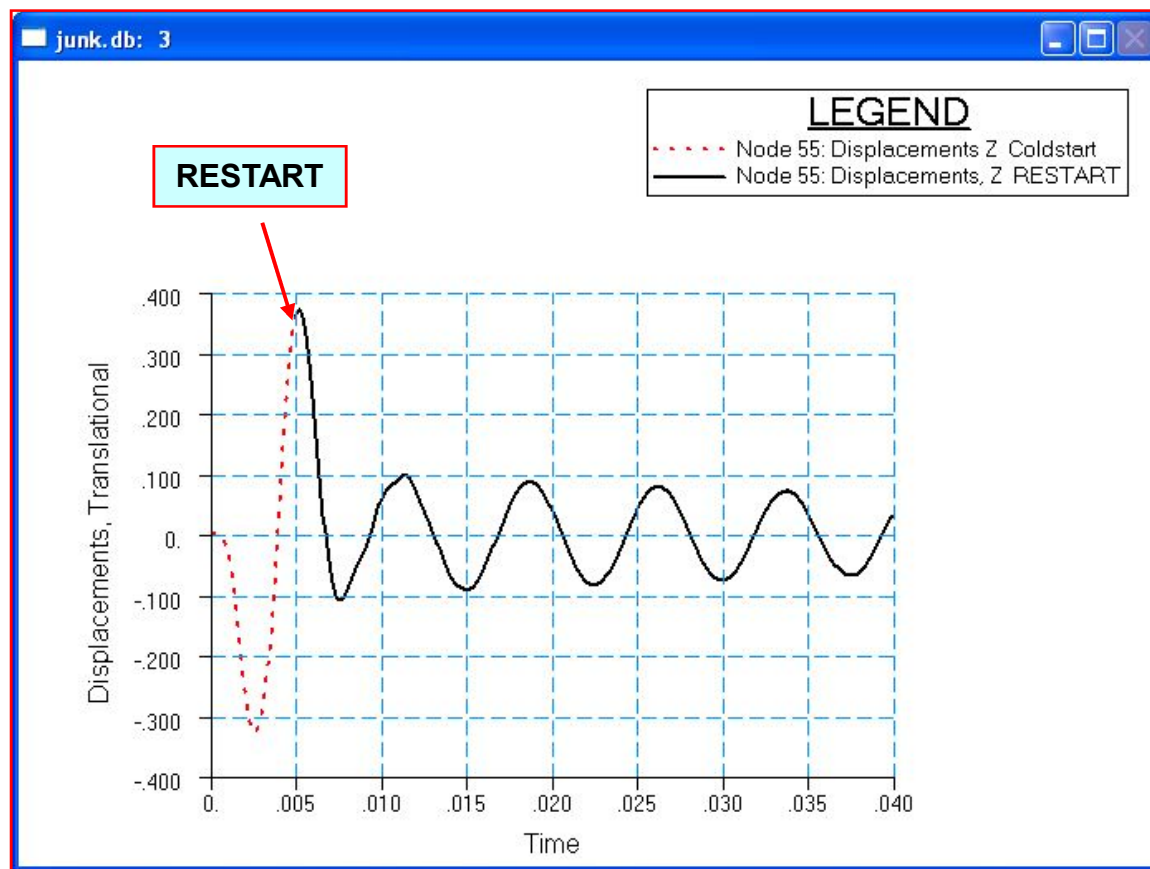
Cold Start Output:

POINT-ID =		D I S P L A C E M E N T V E C T O R						
TIME	TYPE	T1	T2	T3	R1	R2	R3	
0.0	G	0.0	0.0	0.0	0.0	0.0	0.0	
1.000000E-04	G	0.0	0.0	5.377693E-05	2.101296E-04	7.267312E-05	0.0	
...								
4.900000E-03	G	0.0	0.0	3.510944E-01	2.766599E-02	-8.461403E-02	0.0	
5.000000E-03	G	0.0	0.0	3.625650E-01	2.753972E-02	-8.780407E-02	0.0	

Restart Output:

POINT-ID =		D I S P L A C E M E N T V E C T O R						
TIME	TYPE	T1	T2	T3	R1	R2	R3	
5.000000E-03	G	0.0	0.0	3.625650E-01	2.753972E-02	-8.780407E-02	0.0	
5.100000E-03	G	0.0	0.0	3.690938E-01	2.708996E-02	-8.986547E-02	0.0	
...								
3.990000E-02	G	0.0	0.0	2.752052E-02	-6.267783E-05	-7.685922E-03	0.0	
4.000000E-02	G	0.0	0.0	3.238480E-02	-7.384567E-05	-9.044440E-03	0.0	

Transient Response - Restarts



Frequency Response

FREQUENCY = 9.000000E-01

C O M P L E X

POINT ID.	TYPE	T1	T2
0	11	G	0.0
			0.0
0	12	G	-6.682744E-08
			-1.000000E+00

FREQUENCY = 1.000000E+00

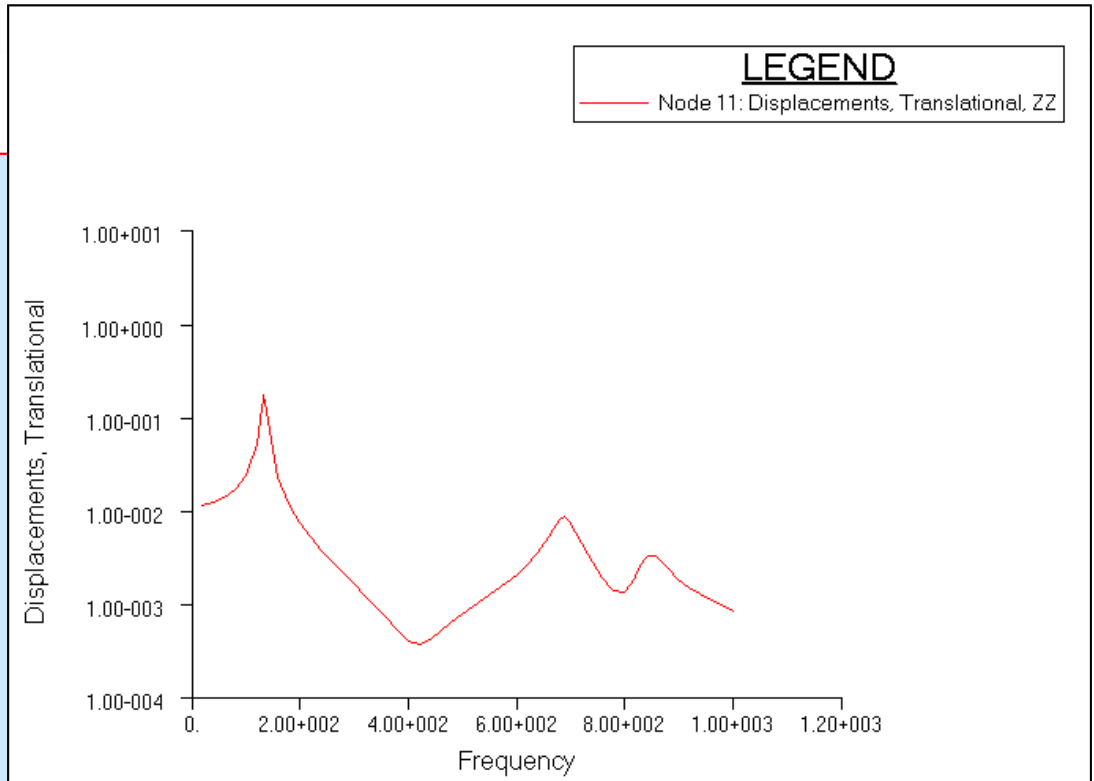
C O M P L E X

POINT ID.	TYPE	T1	T2
0	11	G	0.0
			0.0
0	12	G	-1.046835E-07
			-9.999999E-01

FREQUENCY = 1.100000E+00

C O M P L E X D I S P L A C E M E N T V E C T O R
(REAL/IMAGINARY)

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
0	11	G	0.0	0.0	0.0	0.0	0.0
			0.0	0.0	0.0	0.0	0.0
0	12	G	-6.855670E-08	0.0	0.0	0.0	0.0
			-9.999998E-01	0.0	0.0	0.0	0.0



Frequency Response

- **Compute the response to oscillatory excitation.**
- **Excitation explicitly defined in the frequency domain - all of the applied forces are known at each forcing frequency.**
- **Computed response usually includes nodal displacements and element forces and stresses.**
- **The computed responses are the complex numbers defined as magnitude and phase (with respect to the forcing) or as real and imaginary components.**
- **Two categories of analysis - direct and modal.**

Frequency Response – Direct Method

- **Dynamic equation of motion:**

$$[-\omega^2 \mathbf{M} + i\omega \mathbf{B} + \mathbf{K}]\{\mathbf{u}(\omega)\} = \{\mathbf{P}(\omega)\}$$

- **PARAM,G and GE on MATi entry do not form a damping matrix. They form a complex stiffness matrix**

$$\mathbf{K} = (1 + i\mathbf{G})\mathbf{K}^1 + i\sum \mathbf{G}_E \mathbf{k}_E$$

Where,

$\mathbf{K}^1$ = global stiffness matrix

$\mathbf{G}$ = overall structural damping coefficient (PARAM,G)

$\mathbf{K}_E$ = element stiffness matrix

$\mathbf{G}_E$ = element structural damping coefficient (GE on MATi entry)

- **Contrast this with transient response analysis**

$$\mathbf{B}_{\text{TRANS}} = \mathbf{B}^1 + \mathbf{B}^2 + \frac{\mathbf{G}}{\omega_3} \mathbf{K}^1 + \frac{1}{\omega_4} \sum \mathbf{G}_E \mathbf{k}_E$$

- **Solve the equation by inserting ω to form a complex left-hand side, and then solve it similar to a static problem (using complex arithmetic).**

Frequency Response – Modal Method

- **Convert to modal coordinates and solve as decoupled SDOF systems**

$$\xi_i = \frac{P_i}{-m_i \omega^2 + i b_i \omega + k_i}$$

- **Much quicker to solve this equation than in direct method**
- **Decoupled procedure can be used only if either no damping is present or if modal damping alone (via TABDMP1) is used. Otherwise, use the less efficient direct approach (on smaller modal coordinate matrices) if non-modal damping (VISC, DAMP) is present.**

Frequency Response – Excitation Definition

- **First, you must define force as a function of frequency.**
- **Methods in MSC Nastran:**
 - RLOAD1 (defines frequency-dependent load in real and imaginary forms)
 - RLOAD2 (defines frequency-dependent load in magnitude and phase forms)
- **DLOAD Bulk Data entries are used to combine frequency-dependent forces.**
- **RLOADi entries are selected by DLOAD Case Control commands.**

Frequency Response – Excitation Definition

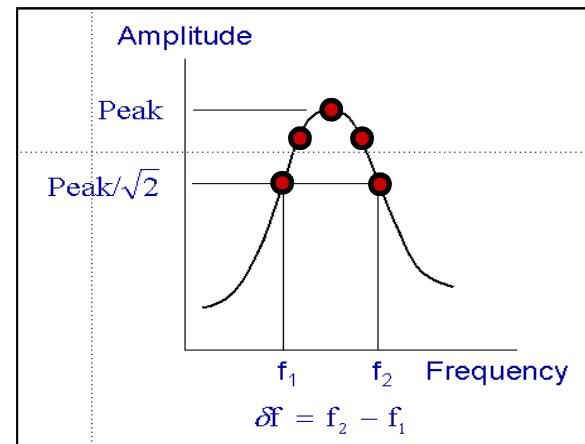
- **Second, you must define the frequencies for excitation.**
- **FREQi Bulk Data entries are selected by the FREQUENCY Case Control commands. All FREQi entries with the same ID are used.**
- **These entries define a list of forcing frequencies to be analyzed.**
- **There will be a corresponding solution at each excitation frequency.**

Frequency Response – Excitation Definition

- **Means of defining the forcing frequency:**
 - **FREQ** - defines discrete excitation frequencies.
 - **FREQ1** - defines f_{START} , frequency increment, and number of increments.
 - **FREQ2** - defines f_{START} , f_{END} , and the number of logarithmic intervals.
 - **FREQ3** - defines $F1$, $F2$, and the number of frequencies in between using either a linear or log interpolation. Biased towards end points or center.
 - **FREQ4** - specifies a frequency at each resonant frequency and the number of equally spaced excitation frequencies within the spread.
 - **FREQ5** - specifies a frequency range and fractions of the natural frequencies within that range.
- The **FREQ3**, **FREQ4**, and **FREQ5** entries are available only for the modal method.

Frequency Response – Considerations

- Exciting an undamped (or modal damped) system at 0.0 Hz gives the same results as a static analysis. Therefore, if the maximum excitation frequency is much less than the lowest resonant frequency of the system, a static analysis is sufficient.
- Very lightly-damped structures exhibit large dynamic responses for excitation frequencies near resonant frequencies. A small change in the model (or running it on another computer) may give large changes in such response.
- Use a fine-enough frequency step size (Δf) to adequately predict peak response. Use at least 5 points per half-power bandwidth.
- For maximum efficiency, use an uneven frequency step size. Use smaller Δf in regions around resonant frequencies and larger Δf in regions far away from resonant frequencies.



Frequency Response – Frequency Dependent Springs/Dampers

- **Specifies stiffness as a function of forcing frequency**
- **Specifies damping as a function of forcing frequency**
- **Different impedance in different direction**
- **CBUSH**
 - Defines generalized spring, damper connectivity
- **PBUSH**
 - Defines nominal spring and damper values
- **PBUSHT**
 - Defines frequency-dependent spring and damper values

Frequency Response - Recommendations

- **Recommendations on FREQ*i* entries**
 - The following combinations are recommended, but not exhaustive:
 - For Modal Method
 - FREQ to specify breakpoints in the force vs frequency curve, as well as in any PSD's
 - FREQ1 set to give the smallest Delta f practical across range of interest
 - FREQ4 Spread set to ζ , %critical damping level, and number to at least 5 to give 5 points evenly spread per half-power bandwidth (use more if higher damping)
 - PARAM,DFREQ set to appropriate filter value
 - For Direct Method
 - FREQ1 set to give the smallest Delta f practical across range of interest
 - FREQ set to all known natural frequencies and % of them to capture data in the half-power bandwidth
 - FREQ to specify breakpoints in the force vs frequency curve, as well as in any PSD's
 - Note:
 - The Direct Method relies heavily on knowledge of the natural frequencies and will be compromised if a single FREQ1 entry alone is used.
 - A common mistake is to miscalculate the range of interest and use FREQ1 only– and get a null result, or incomplete results

OUTPUT(XYPLOT)

- **Old method of getting data recovery out of Nastran**
 - Generally, it is preferable to use a post-processor to view results (using the op2 or XDB), but for historical reasons and some required applications, this method is still available
- **Many options for creating plots of the model and results**
 - Will not go over everything as it is too involved for this class
- **This goes at the end of the Case Control and before Begin Bulk**
- **Example to create a graph of the displacement response of Grid 1 in the T2 DOF:**

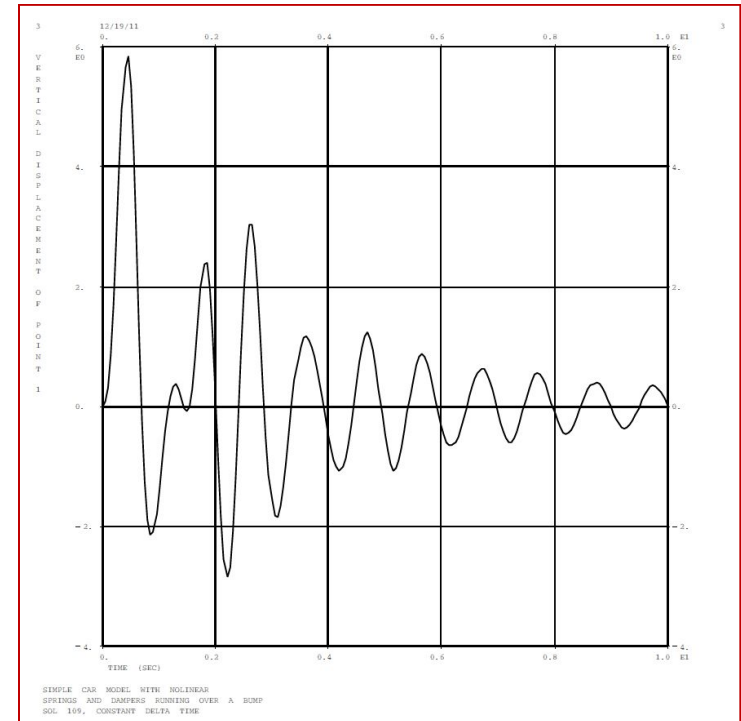
```
OUTPUT(XYPLOT)
CSCALE=1.3
XAXIS= YES
YAXIS= YES
XGRID LINES= YES
YGRID LINES= YES
XTITLE= TIME (SEC)
YTITLE= VERTICAL DISPLACEMENT OF POINT 1
XYPLOT DISP/1(T2)
```

OUTPUT(XY PLOT)

- **The default output resulting from use of the Nastran plotter is a .plt file**
 - This is a binary file
 - Unless you send this file to a plotter, it needs conversion for viewing
 - The conversion is done using PLOTPS (discussed on the next slide)
- **Alternately, the .ps file can be created automatically by adding the following line to the FMS section of the Nastran input file:**

ASSIGN PLOT=*,FORM=FORMATTED

- This has been done for the only 2 Workshops using this method, (Workshop 3 and Workshop 7)
- The ASSIGN can be added to the system rc file and then will be the default for all runs



An example of a .ps file

PLOTPS

- **Utility provided with the MSC Nastran installation to convert *.plt files to *.ps files for viewing with any postscript viewer (such as "ghostview" which is freeware and should be available on the net)**
- **The utility can be found with the MSC Nastran installation in a location like: C:\MSC.Software\MSC_Nastran\20130\msc20130\win64\plotps.exe**
- **The easiest method for using this is to double-click on the *.plt file in a Windows folder, and when it comes up asking what program to use to open, select browse and point to the plotps.exe**
 - From that point onward, just double-clicking on the *.plt file will convert it to a *.ps file
- **This is only documented here for historical purposes, but is no longer required, as adding "ASSIGN PLOT=*, FORMATTED" to the input file is much simpler**

SECTION 3

Dynamic Analysis Checkout Tools

Items to Review

- **Miscellaneous Model Checkout Tools**
 - Estimating Fundamental Natural Frequency
 - Animation for Mode Identification
 - Groundcheck
 - Rigid Body Modes
- **Modal Effective Mass**
- **Using Strain Energy to Identify Areas for Change**
- **Grid Point Kinetic Energy**
- **Weight Check**
- **Mode Selection**
- **Modal Contribution Factor**

Miscellaneous Model Checkout Tools

Estimating Fundamental Natural Frequency

Estimating Fundamental Natural Frequency

- **The correct natural frequency for a structure may not be known... how do you gain confidence in your results?**
 - Handbooks providing closed form solutions (only good for simple structures such as plates and beams)
 - Historical data within the company
 - Idealizations to a simple structure, then use handbooks
- **If you want to get a rough estimate of the natural frequency of a more complex structure, use the following equation:**

$$f = \frac{1}{2\pi} \sqrt{\frac{g}{d}}$$

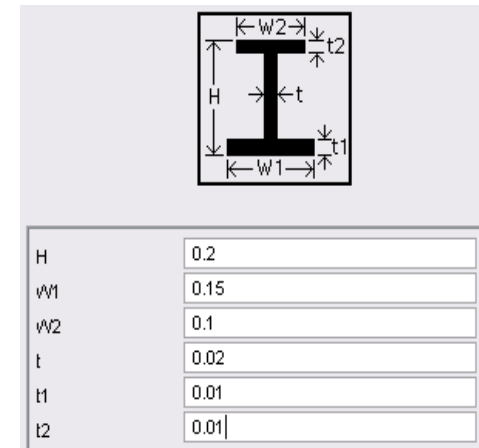
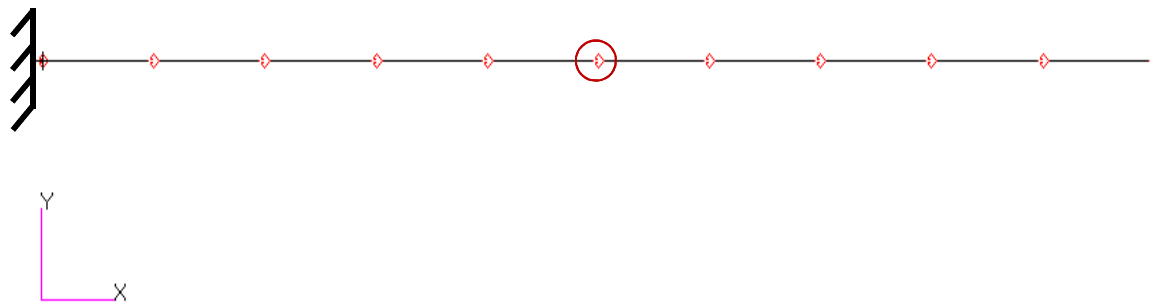
where:

d = center of gravity location displacement in the direction of an applied 1g-load

g = acceleration due to gravity (in the model units)

Estimating Fundamental Natural Frequency (cont.)

- The following sample demonstrates this point
 - $L=10\text{in}$, $E=30\text{E}6$, $\text{Nu}=.3$, $\text{Rho}=7.4\text{E-}4$ and cross section as shown



- The center grid is GRID 6, and the displacement results of running a static analysis with 386.4 in/sec^2 applied in each axis is shown on the next slide
 - Using sample file beam_sol101-grav.bdf

Estimating Fundamental Natural Frequency (cont.)

DISPLACEMENT VECTOR								SUBCASE 1
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
6	G	3.574200E-07	3.236052E-21	-1.190630E-24	0.0	4.003881E-25	1.561341E-21	
DISPLACEMENT VECTOR								SUBCASE 2
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
6	G	1.108310E-05	8.095533E-04	-2.857025E-19	-6.411090E-09	9.414526E-20	2.662731E-04	
DISPLACEMENT VECTOR								SUBCASE 3
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
6	G	1.487532E-09	-2.857052E-19	6.853668E-03	4.050528E-04	-2.257938E-03	-9.414523E-20	

Estimating Fundamental Natural Frequency (cont.)

- Running a normal modes analysis of the same model results in the following eigenvalues
 - Using sample file beam_sol103_30modes.bdf

MODE NO.	EXTRACTION ORDER	EIGENVALUE	REAL EIGENVALUES		GENERALIZED MASS	GENERALIZED STIFFNESS
			RADIANS	CYCLES		
1	1	3.061045E+04	1.749584E+02	2.784549E+01	1.000000E+00	3.061045E+04
2	2	2.596890E+05	5.095969E+02	8.110487E+01	1.000000E+00	2.596890E+05
3	3	1.163624E+06	1.078714E+03	1.716827E+02	1.000000E+00	1.163624E+06
4	4	6.278638E+06	2.505721E+03	3.987979E+02	1.000000E+00	6.278638E+06
23	23	9.982427E+08	3.159498E+04	5.028497E+03	1.000000E+00	9.982427E+08
27	27	1.771987E+09	4.209497E+04	6.699623E+03	1.000000E+00	1.771987E+09
28	28	2.763933E+09	5.257312E+04	8.367272E+03	1.000000E+00	2.763933E+09
29	29	4.366275E+09	6.607780E+04	1.051661E+04	1.000000E+00	4.366275E+09
30	30	6.073054E+09	7.792980E+04	1.240291E+04	1.000000E+00	6.073054E+09

- The hand calculated prediction compares well with the finite element solution
 - For the x-direction, $f = (1/2\pi) * (386.4/3.574e-7)^{1/2} = 5233.1$ Hz

Mode direction	prediction	From FEM
Fx	5233 Hz	5028 Hz
Fy	110 Hz	81 Hz
Fz	38 Hz	28 Hz

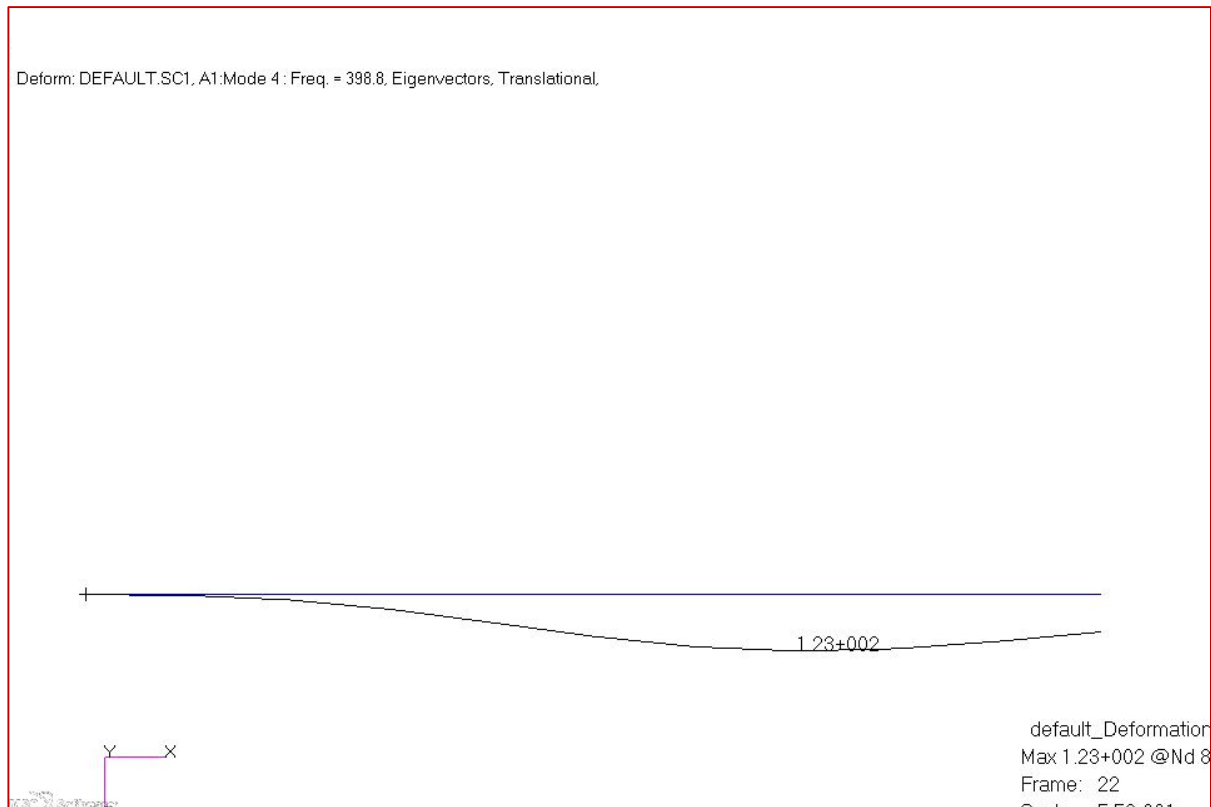
Animation for Mode Identification

Animation for Mode Identification

- **One of the best tools for characterizing a mode is visualization**
- **Animation allows for the most effective means of identification, and the ability to rotate the display during animation allows for observing motion that might be obscured**
- **There are a couple conditions in which an animation can be misleading**
 - A point mass rotating
 - A line element torsioning about its own axis
- **In these situations, it is possible for a post-processor to amplify noise in the response and provide an inappropriate display**
- **Good modeling techniques eliminate this risk**
 - Make sure all lines and point masses have some cross-sectional connectivity

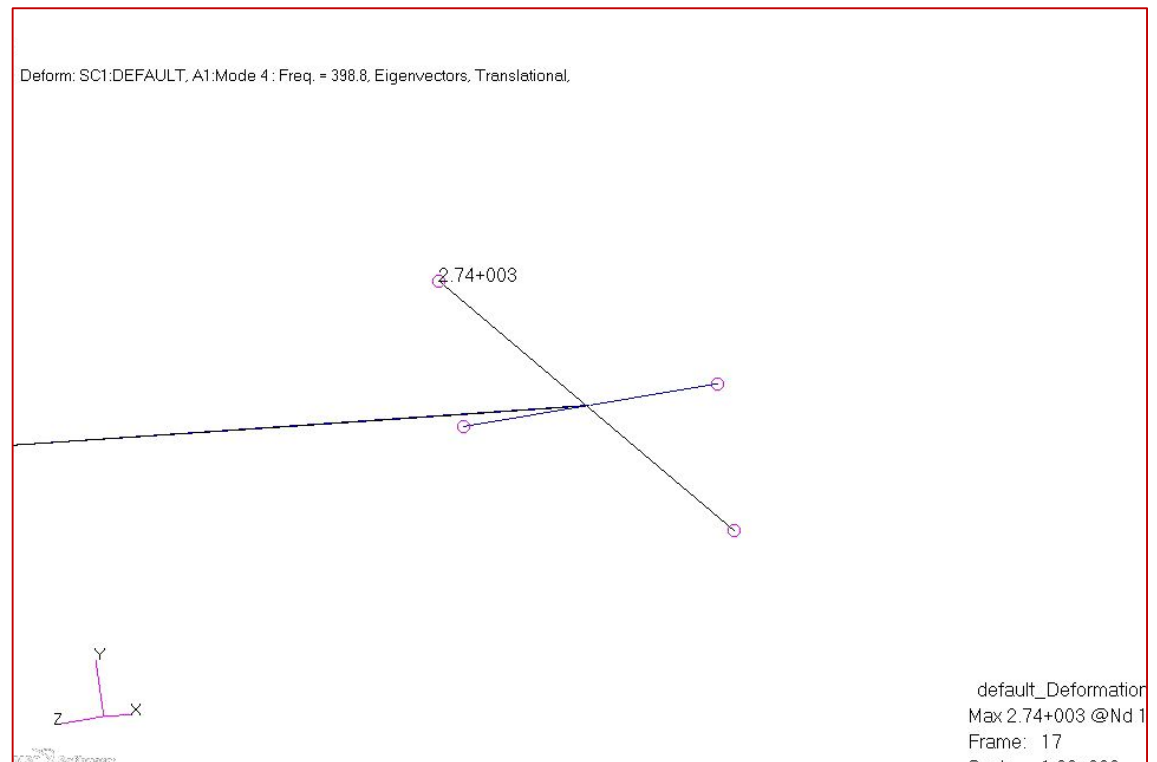
Animation for Mode Identification (cont.)

- For example, using the beam model in the previous section, the 4th mode, when deformed gives the following type of motion (undeformed is blue, deformed is black) Sample file: beam_sol103_10modes_no-xsection.bdf



Animation for Mode Identification (cont.)

- Using some of the tools discuss later in this section, it will be shown this mode is really a torsional mode.
- Adding additional structure (using dummy grids and an RBE2) resulted in the this deformation (using file beam_sol103_10modes.bdf):
- There is some very small out-of-plane bending, which, lacking any other information to display, was amplified in the previous deformation. The deformation here is seen in a more appropriate context with the majority of motion represented by the beam torsioning.



Grounding Check & Rigid Body Modes

Grounding Checks

- **GROUNDCHECK helps identify grounding problems in the model.**
 - One of the biggest advantages of this capability is to evaluate the model at various stages of processing.
 - This helps to identify at what stage the grounding is introduced, helping to pinpoint the source of error.
- **While not specific to normal modes (it does not require mass to calculate) it is available in the normal modes solution.**
- **A grounding check is also referred to as a rigid body check because the stiffness matrix is multiplied by the rigid body transformation matrix.**
 - The grounding check identifies unintentional constraints and ill-conditioning in the stiffness matrix.

Grounding Checks (cont.)

- The grounding check is requested with the **GROUNDCHECK Case Control** command.

GROUNDCHECK

Rigid Body Motion Grounding Check

Perform grounding check analysis on stiffness matrix to expose unintentional constraints by moving the model rigidly.

Format:

$$\text{GROUNDCHECK} \left[\begin{array}{l} \left(\begin{array}{l} \text{PRINT} \\ \text{NOPRINT} \end{array} \right), \text{PUNCH, SET} = \left(\begin{array}{l} \text{G, N, N + AUTOSPC, F, A} \\ \text{ALL} \end{array} \right) \\ \text{GRID} = \text{gid, THRESH} = \text{e, DATA REC} = \begin{array}{l} \text{YES} \\ \text{NO} \end{array}, (\text{RTHRESH} = \text{r}) \end{array} \right] = \left\{ \begin{array}{l} \text{YES} \\ \text{NO} \end{array} \right\}$$

Grounding Checks (cont.)

- **User Information Message 7570 is issued by GROUNDCHECK for each DOF set requested.**
 - The strain energy is computed in each direction of the rigid body motion and by default, if the strain energy exceeds the tolerance, then “FAIL” is printed out for that direction.
 - The tolerance is set by dividing the largest stiffness term by 1.E10.
 - This default value can be changed with the THRESH keyword
 - The RTHRESH keyword can be used to further control the amount of output.
 - Possible reasons for failure are also printed after the strain energies.
- **It is recommended that SET=ALL be used to enable evaluation at all processing levels in the solution (GSET, NSET, FSET, ASET, etc).**

Grounding Checks (cont.)

- From the sample beam model (beam_sol103_10modes.bdf), the check indicates passing for the G-Set.
 - As shown in the informational message, common reasons for failing at this level are CELAS elements connected to only one grid, or connected between non-coincident grids or improper DMIG matrices.

```
*** USER INFORMATION MESSAGE 7570 (GPWG1D)
RESULTS OF RIGID BODY CHECKS OF MATRIX KGG      (G-SET) FOLLOW:
PRINT RESULTS IN ALL SIX DIRECTIONS AGAINST THE LIMIT OF 3.660000E-05
      DIRECTION      STRAIN ENERGY      PASS/FAIL
      -----
        1      0.000000E+00      PASS
        2      0.000000E+00      PASS
        3      0.000000E+00      PASS
        4      0.000000E+00      PASS
        5      1.330136E-11      PASS
        6      5.184120E-11      PASS

SOME POSSIBLE REASONS MAY LEAD TO THE FAILURE:
  1. CELASI ELEMENTS CONNECTING TO ONLY ONE GRID POINT;
  2. CELASI ELEMENTS CONNECTING TO NON-COINCIDENT POINTS;
  3. CELASI ELEMENTS CONNECTING TO NON-COLINEAR DOF;
  4. IMPROPERLY DEFINED DMIG MATRICES;
  5. CBEAM/CBEAM3 WITH GRID ID FOR WARPING DOF
```

Grounding Checks (cont.)

- Looking at the N-Set, we see this test was passed. This is to be expected, as there are no rigid elements in the model.
 - As indicated in the Information Message, multipoint constraint equations that do not satisfy rigid body motion can cause failure.

```
*** USER INFORMATION MESSAGE 7570 (GPWG1D)
RESULTS OF RIGID BODY CHECKS OF MATRIX KNN      (N-SET) FOLLOW:
PRINT RESULTS IN ALL SIX DIRECTIONS AGAINST THE LIMIT OF 3.660000E-05
DIRECTION      STRAIN ENERGY      PASS/FAIL
-----
1              0.000000E+00      PASS
2              0.000000E+00      PASS
3              0.000000E+00      PASS
4              0.000000E+00      PASS
5              1.330136E-11      PASS
6              5.184120E-11      PASS

SOME POSSIBLE REASONS MAY LEAD TO THE FAILURE:
1. MULTIPOINT CONSTRAINT EQUATIONS WHICH DO NOT SATISFY RIGID-BODY MOTION;
2. RBE3 ELEMENTS FOR WHICH THE INDEPENDENT DEGREE-OF-FREEDOM CANNOT DESCRIBE
ALL POSSIBLE RIGID-BODY MOTIONS.
```

Grounding Checks (cont.)

- **Looking at the F-Set, we see this test was failed.**
 - As indicated in the Information Message, constraints in the model can cause failure, which is the case for the beam model.
 - So, even the fact that this failed is a good model check... had it not failed all 6 DOF, then there would have been something wrong with the intended constraints.

```
*** USER INFORMATION MESSAGE 7570 (GPWG1D)
RESULTS OF RIGID BODY CHECKS OF MATRIX KFF      (F-SET)  FOLLOW:
PRINT RESULTS IN ALL SIX DIRECTIONS AGAINST THE LIMIT OF  3.660000E-05
DIRECTION      STRAIN ENERGY      PASS/FAIL
-----
1              9.150000E+04        FAIL
2              4.467594E+03        FAIL
3              6.424539E+02        FAIL
4              3.360642E+00        FAIL
5              2.171010E+02        FAIL
6              1.754422E+03        FAIL

SOME POSSIBLE REASONS MAY LEAD TO THE FAILURE:
1. CONSTRAINTS WHICH PREVENT RIGID-BODY MOTION.
```

Grounding Checks (cont.)

- **Looking at the A-Set, we see this test also failed.**
 - This is to be expected, as the ASET is the same as the FSET in this model.
 - As indicated in the Information Message, constraints in the model can cause failure, which is the case for the beam model.
 - Additional potential numerical causes are also listed.

```
*** USER INFORMATION MESSAGE 7570 (GPWG1D)
RESULTS OF RIGID BODY CHECKS OF MATRIX KAA1      (A-SET) FOLLOW:
PRINT RESULTS IN ALL SIX DIRECTIONS AGAINST THE LIMIT OF 3.660000E-05
DIRECTION      STRAIN ENERGY      PASS/FAIL
-----
1              9.150000E+04        FAIL
2              4.467594E+03        FAIL
3              6.424539E+02        FAIL
4              3.360642E+00        FAIL
5              2.171010E+02        FAIL
6              1.754422E+03        FAIL

IF THE MODEL HAS PASSED THE PREVIOUS CHECKS FOR THE G-SET AND N-SET,
THEN SOME POSSIBLE CAUSES ARE:
1. THE MODEL IS NOT INTENDED TO BE FREE-FREE WHICH INDICATES THAT THE MODEL IS
   PROPERLY CONSTRAINED TO GROUND;
2. THE REFERENCE GRID POINT (GRID=GID ON THE GROUNDCHECK COMMAND) IS LOCATED
   TOO FAR FROM THE MODEL'S CENTER OF GRAVITY. IT IS RECOMMENDED THAT THE
   REFERENCE GRID POINT BE LOCATED AS CLOSE AS POSSIBLE TO THE MODEL'S CENTER
   OF GRAVITY OF THE MODEL (SEE THE GRID POINT WEIGHT GENERATOR OUTPUT);
3. PARAM,AUTOSPC,YES CONSTRAINS NEAR-SINGULAR DEGREES-OF-FREEDOM. WHEN A
   FINITE ELEMENT MODEL WITH AUTOSPC FAILS THE A-SET CHECK, IT IS NOT EVIDENT
   THAT GROUNDING HAS OCCURRED. THE USE OF PARAM,SNORM WILL NOT ELIMINATE THE
   SPURIOUS FAILURE.
```

Grounding Checks (cont.)

- **If the DATAREC keyword is specified and the rigid body check fails, then data recovery will be performed to compute and print the grounding forces to make it easier to locate the source of the failure.**
 - By default, only those grounding forces larger than 10 percent of the largest grounding force will be printed (see RTHRESH keyword).
- **Since data recovery can only be performed on DOF that are part of the set being evaluated, the grid showing the forces will often be ‘near’ the real grid.**
 - In the beam example, since grid 1 was constrained, that is the grid that should show up with the grounding forces. But when a grid is constrained, it is removed for the solution set, so for the F-Set, grid 1 doesn’t exist. So grid 2 shows the grounding force.

Grounding Checks (cont.)

GROUND CHECK FORCES (F - S E T)								DIRECTION	1
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3		
2	G	1.830000E+05	0.0	0.0	0.0	0.0	0.0		
GROUND CHECK FORCES (F - S E T)								DIRECTION	2
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3		
2	G	0.0	8.935188E+03	0.0	0.0	0.0	-4.467594E+03		
GROUND CHECK FORCES (F - S E T)								DIRECTION	3
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3		
2	G	0.0	0.0	1.284908E+03	0.0	6.424539E+02	0.0		
GROUND CHECK FORCES (F - S E T)								DIRECTION	4
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3		
2	G	0.0	0.0	0.0	6.721284E+00	0.0	0.0		
GROUND CHECK FORCES (F - S E T)								DIRECTION	5
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3		
2	G	0.0	0.0	-6.424539E+02	0.0	-2.082519E+02	0.0		
GROUND CHECK FORCES (F - S E T)								DIRECTION	6
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3		
2	G	-7.617019E+03	4.467594E+03	0.0	0.0	0.0	-9.587498E+02		

Rigid Body Modes

Rigid Body Modes

- **When performing a normal modes analysis, even if the structure is intended to be constrained, a free-free modes run should be performed.**
- **It is important to verify:**
 - There are the expected number of rigid body modes (no more/no less)
 - The rigid body modes are near zero in frequency
 - There is good separation between the highest rigid body mode and the lowest elastic mode (4-6 orders of separation)

Rigid Body Modes (cont.)

- The beam model (sample file beam_sol103_10modes-freefree.bdf) was run and the eigenvalue table shown below
- As expected, there are 6 rigid body modes
 - If there were more, it could indicate a mechanism or unconnected components
 - If there were less, it could indicate grounding or unintended constraints
- There are 6 orders of magnitude separation between the largest rigid body mode and the lowest elastic mode

MODE NO.	EXTRACTION ORDER	EIGENVALUE	R E A L E I G E N V A L U E S		GENERALIZED MASS	GENERALIZED STIFFNESS
			RADIANS	CYCLES		
1	1	-2.279878E-06	1.509926E-03	2.403122E-04	1.000000E+00	-2.279878E-06
2	2	-3.790483E-07	6.156689E-04	9.798675E-05	1.000000E+00	-3.790483E-07
3	3	-3.864989E-08	1.965957E-04	3.128919E-05	1.000000E+00	-3.864989E-08
4	4	-1.164153E-08	1.078959E-04	1.717217E-05	1.000000E+00	-1.164153E-08
5	5	5.122274E-09	7.157007E-05	1.139073E-05	1.000000E+00	5.122274E-09
6	6	1.024455E-08	1.012154E-04	1.610892E-05	1.000000E+00	1.024455E-08
7	7	1.174406E+06	1.083700E+03	1.724763E+02	1.000000E+00	1.174406E+06
8	8	8.466842E+06	2.909784E+03	4.631065E+02	1.000000E+00	8.466842E+06
9	9	9.970671E+06	3.157637E+03	5.025535E+02	1.000000E+00	9.970671E+06
10	10	2.496359E+07	4.996358E+03	7.951950E+02	1.000000E+00	2.496359E+07

Modal Effective Mass

Modal Effective Mass

- **Accuracy in a dynamics solution is greatly affected by the mass**
- **Insight into the modal characteristics of the structure being analyzed can be aided**
 - By identifying the direction the mass in a mode is moving
 - By identifying how much mass each mode contributes to the total mass
- **Until an actual load is applied, it is difficult to predict which modes will contribute the most to the response**
- **Modal effective mass can help overcome this deficiency**

Modal Effective Mass (cont.)

- **Linear combinations of eigenvectors can be used to define any vector**
 - Eigenvectors calculated in a normal modes analysis are linearly independent of each other and span the vector space used to define the model response
 - A rigid body vector can be constructed from a set of flexible body eigenvectors, with it having motion in a desired direction
- **The relationship between the rigid body vector $\{D\}_R$ and the corresponding rigid body mass, M_R , is**

$$M_R = \{D\}_R^T [M] \{D\}_R$$

Using $\{D\}_R = [\phi] \{\varepsilon\}$

Then $M_R = \{\varepsilon\}^T [\phi]^T [M] [\phi] \{\varepsilon\} = \{\varepsilon\}^T [m] \{\varepsilon\}$

- **The contribution which the i-th mode provides to the rigid body mass M_R is $e_i^2 m_{ii}$. This is known as the modal effective mass.**

Modal Effective Mass (cont.)

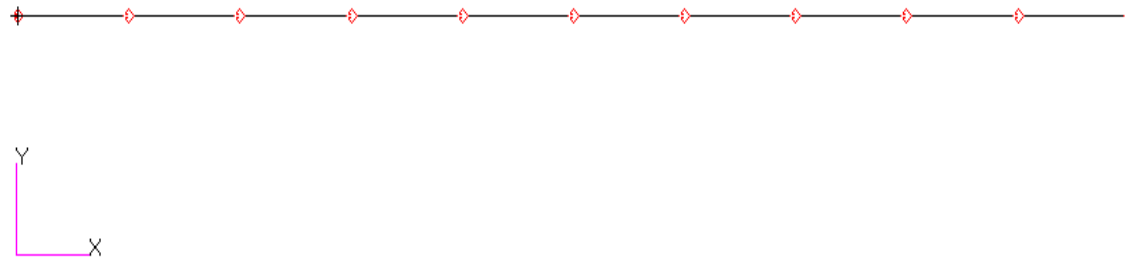
- **The values calculated are used by different industries in different ways:**
 - For example, in Civil Engineering seismic analysis, the contribution from each mode is assessed as a percentage and the total is summed. Any shortfall from 100% is classified as 'missing mass'.
 - If the missing mass is significant, then it may indicate errors in the analysis, typically insufficient modes being used in a modal method.
 - 'Missing mass' is often characterized as higher frequency body type loading and can be simulated by applying a 1g inertia load in the appropriate direction, factored by the % missing mass, then added as a static load.
- **In testing, many times, the modal effective mass is used as a way of measuring whether sufficient modes have been retained.**

Modal Effective Mass (cont.)

- The **MEFFMASS** case control command can provide this information

$$\text{MEFFMASS} \left[\begin{array}{c} \text{PRINT} \\ \text{NOPRINT} \end{array} \right] \left[\begin{array}{c} \text{PUNCH} \\ \text{NOPUNCH} \end{array} \right], \text{GRID} = \text{gid}, \left[\begin{array}{c} \text{SUMMARY, PARTFAC,} \\ \text{MEFFM, MEFFW,} \\ \text{FRACSUM, ALL} \end{array} \right] = \left\{ \begin{array}{c} \text{YES} \\ \text{NO} \end{array} \right\}$$

- **Let's look at the output found in the f06 file for a eigensolution solving for the first 10 modes of a cantilevered beam.**
 - Using sample file beam_sol103_10modes.bdf



Modal Effective Mass (cont.)

- Effective Mass Matrix is the 6x6 mass represented by the modes, while the A-SET Rigid Body Mass Matrix is the actual 6x6 rigid body mass of the structure
- Total Effective Mass Fraction printout shows how much of the total possible rigid body mass is represented by the modes

```

TOTAL EFFECTIVE MASS FRACTION
REFERENCE POINT AT ORIGIN OF BASIC COORDINATE SYSTEM

      T1      T2      T3      R1      R2      R3
9.658041E-18  9.106091E-01  9.657330E-01  9.544342E-01  9.996986E-01  9.985170E-01

EFFECTIVE MASS MATRIX

***
*  4.141657E-22  3.485905E-14  2.779616E-16  8.022988E-17 -1.545851E-16  4.482227E-14 *
*  3.485905E-14  3.904965E-05 -3.352708E-12 -2.738033E-11  2.219285E-12  2.228940E-04 *
*  2.779616E-16 -3.352708E-12  4.141353E-05  1.762739E-06 -2.248915E-04 -2.502440E-12 *
*  8.022988E-17 -2.738033E-11  1.762739E-06  3.104392E-07 -9.389909E-06 -1.484758E-10 *
* -1.545851E-16  2.219285E-12 -2.248915E-04 -9.389909E-06  1.511734E-03  1.464653E-12 *
*  4.482227E-14  2.228940E-04 -2.502440E-12 -1.484758E-10  1.464653E-12  1.510022E-03 *
***

A-SET RIGID BODY MASS MATRIX

***
*  4.288300E-05  0.000000E+00  0.000000E+00  0.000000E+00  2.825136E-11 -1.784921E-06 *
*  0.000000E+00  4.288300E-05  0.000000E+00 -2.825136E-11  0.000000E+00  2.257000E-04 *
*  0.000000E+00  0.000000E+00  4.288300E-05  1.784921E-06 -2.257000E-04  0.000000E+00 *
*  0.000000E+00 -2.825136E-11  1.784921E-06  3.252599E-07 -9.394323E-06 -1.486914E-10 *
*  2.825136E-11  0.000000E+00 -2.257000E-04 -9.394323E-06  1.512190E-03 -1.175908E-12 *
* -1.784921E-06  2.257000E-04  0.000000E+00 -1.486914E-10 -1.175908E-12  1.512264E-03 *
***

```


Modal Effective Mass (cont.)

- In the T1 direction, no mass is represented. This means there are no modes in which the mass of the structure is moving in the T1 direction.
 - It would be difficult to dynamically excite this structure in the T1 direction, and any applied loading in that direction would likely result in no response.
- In the T2 and T3 directions, 91.1% and 96.6% of the possible mass is represented by the first 10 modes.

TOTAL EFFECTIVE MASS FRACTION
REFERENCE POINT AT ORIGIN OF BASIC COORDINATE SYSTEM

T1	T2	T3	R1	R2	R3
9.658041E-18	9.106091E-01	9.657330E-01	9.544342E-01	9.996986E-01	9.985170E-01

Modal Effective Mass (cont.)

- If the eigensolution solves for 30 modes instead of 10, then 85% of the available mass is represented in the T1 direction, while the T2 and T3 directions improve to 100%.

TOTAL EFFECTIVE MASS FRACTION					
REFERENCE POINT AT ORIGIN OF BASIC COORDINATE SYSTEM					
T1	T2	T3	R1	R2	R3
8.497244E-01	9.990354E-01	1.000000E+00	1.000000E+00	1.000000E+00	9.999877E-01

- In addition to the Modal Effective Mass Summary, the Mass Fraction for each mode is provided.

Modal Effective Mass (cont.)

- In our BEAM example, the original visual inspection of the mode shapes indicated mode 4 to be a form of bending mode with deformation in the T3 direction.
- Upon evaluating the translational modal effective mass fractions we see mode 4 as having no motion in the T1 and T2 direction and only .04% of the mass moving in the T3 direction.

MODAL EFFECTIVE MASS FRACTION (FOR TRANSLATIONAL DEGREES OF FREEDOM)							
MODE NO.	FREQUENCY	FRACTION	T1 SUM	FRACTION	T2 SUM	FRACTION	T3 SUM
1	2.784549E+01	1.831535E-37	1.831535E-37	5.009826E-16	5.009826E-16	6.432269E-01	6.432269E-01
2	8.110487E+01	1.702538E-33	1.702721E-33	6.431246E-01	6.431246E-01	1.646685E-16	6.432269E-01
3	1.716827E+02	1.979080E-38	1.702741E-33	1.533017E-15	6.431246E-01	1.990311E-01	8.422580E-01
4	3.987979E+02	2.758016E-35	1.730321E-33	8.356104E-12	6.431246E-01	4.098169E-04	8.426679E-01
5	4.732138E+02	2.074802E-36	1.732396E-33	2.246501E-15	6.431246E-01	6.780667E-02	9.104745E-01
6	5.009877E+02	1.246632E-30	1.248365E-30	1.990124E-01	8.421370E-01	1.620061E-14	9.104745E-01
7	9.042602E+02	9.570506E-04	9.570507E-04	1.051761E-12	8.421370E-01	2.474705E-02	9.450016E-01

Modal Effective Mass (cont.)

- As can be seen from the rotational modal effective mass fractions (in the table below), mode 4 has 66.3% of all the mass in the R1 direction.
- This corresponds to rotation about the axis for the beam. This helps to identify mode 4 as a torsional mode, not a bending mode as the original deformation suggested.

MODAL EFFECTIVE MASS FRACTION (FOR ROTATIONAL DEGREES OF FREEDOM)							
MODE NO.	FREQUENCY	FRACTION	R1 SUM	FRACTION	R2 SUM	FRACTION	R3 SUM
1	2.784549E+01	1.484973E-01	1.484973E-01	9.704365E-01	9.704365E-01	6.124483E-16	6.124483E-16
2	8.110487E+01	4.042704E-11	1.484973E-01	1.948501E-16	9.704365E-01	9.703533E-01	9.703533E-01
3	1.716827E+02	5.929715E-02	2.077944E-01	2.488551E-02	9.953220E-01	1.698432E-15	9.703533E-01
4	3.987979E+02	6.633498E-01	8.711443E-01	3.129946E-07	9.953223E-01	5.275875E-13	9.703533E-01
5	4.732138E+02	2.670934E-03	8.738152E-01	3.223243E-03	9.985456E-01	1.148605E-16	9.703533E-01
6	5.009877E+02	2.422992E-12	8.738152E-01	1.835114E-16	9.985456E-01	2.495353E-02	9.953068E-01
7	8.812608E+02	8.461781E-03	8.888608E-01	8.247147E-04	8.888608E-01	1.822788E-15	8.858608E-01

Modal Effective Mass (cont.)

- By reviewing the printout of the full, 30-mode solution, you can see the 23rd mode is required to get T1 mass participation.
- Scan the 'Fraction' column to see how many modes are required to achieve the desired modal mass participation in a particular direction.
 - If 90% modal effective mass were the threshold, more modes are needed for the T1 direction, while the T2 direction achieved the goal by the 9th mode and the T3 direction by the 5th mode.

MODE NO.	FREQUENCY	FRACTION	T1	SUM	FRACTION	T2	SUM	FRACTION	T3	SUM
1	2.784549E+01	0.000000E+00		0.000000E+00	5.009827E-16		5.009827E-16	6.432269E-01		6.432269E-01
2	8.110487E+01	1.745503E-33		1.745503E-33	6.431246E-01		6.431246E-01	1.646685E-16		6.432269E-01
3	1.716827E+02	0.000000E+00		1.745503E-33	1.533017E-15		6.431246E-01	1.990311E-01		8.422580E-01
4	3.987979E+02	1.823780E-38		1.745521E-33	8.356104E-12		6.431246E-01	4.098169E-04		8.422580E-01
5	4.732138E+02	4.451031E-37		1.745966E-33	2.246501E-15		6.431246E-01	6.780667E-02		9.104745E-01
6	5.009877E+02	1.525912E-33		3.271878E-33	1.990124E-01		8.421370E-01	1.620061E-14		9.104745E-01
7	9.043693E+02	8.522276E-37		3.272730E-33	1.251761E-13		8.421370E-01	3.474705E-02		9.452216E-01
8	1.189049E+03	1.194202E-34		3.392150E-33	9.694280E-12		8.421370E-01	3.728940E-05		9.452589E-01
9	1.381379E+03	1.331206E-32		1.670421E-32	6.847201E-02		9.106091E-01	1.513553E-12		9.452589E-01
10	1.455059E+03	1.600904E-34		1.686430E-32	3.830077E-12		9.106091E-01	2.047413E-02		9.657330E-01
11	1.936433E+03	6.623151E-32		8.309581E-32	3.594908E-14		9.106091E-01	1.601773E-03		9.673348E-01
12	2.105677E+03	1.395282E-32		9.704863E-32	3.439524E-14		9.106091E-01	1.134252E-02		9.786773E-01
13	2.612831E+03	4.733818E-32		1.443868E-31	5.509758E-10		9.106091E-01	3.072235E-03		9.817495E-01
14	2.657073E+03	6.414715E-31		7.858583E-31	3.503041E-02		9.456395E-01	9.242968E-11		9.817495E-01
15	2.814736E+03	3.927075E-30		4.712934E-30	4.960295E-11		9.456395E-01	5.159944E-03		9.869095E-01
16	3.195627E+03	6.306938E-30		1.101987E-29	4.428950E-12		9.456395E-01	2.593069E-03		9.895025E-01
17	3.517574E+03	5.813382E-31		1.160121E-29	1.540106E-11		9.456395E-01	2.399540E-03		9.919021E-01
18	3.628481E+03	6.068063E-29		7.228184E-29	2.632713E-14		9.456395E-01	1.583850E-03		9.934859E-01
19	3.881179E+03	5.364428E-31		7.281829E-29	3.304885E-13		9.456395E-01	5.597858E-04		9.940457E-01
20	4.206037E+03	1.069668E-27		1.142486E-27	9.557541E-11		9.456395E-01	1.768778E-03		9.958145E-01
21	4.294399E+03	5.284260E-26		5.398509E-26	2.106229E-02		9.667017E-01	6.386849E-12		9.958145E-01
22	4.861386E+03	3.716141E-25		4.255992E-25	9.260902E-15		9.667017E-01	1.410476E-03		9.972250E-01
23	5.028497E+03	8.497244E-01		8.497244E-01	3.011020E-27		9.667017E-01	7.568943E-28		9.972250E-01
24	5.518394E+03	3.304170E-28		8.497244E-01	1.021861E-12		9.667017E-01	1.342164E-03		9.985672E-01
25	6.183575E+03	4.202096E-30		8.497244E-01	8.412423E-12		9.667017E-01	1.045130E-03		9.996123E-01
26	6.232623E+03	1.254686E-26		8.497244E-01	1.379040E-02		9.804922E-01	5.647632E-13		9.996123E-01
27	6.699623E+03	5.332592E-27		8.497244E-01	3.878656E-14		9.804922E-01	3.877158E-04		1.000000E+00
28	8.367272E+03	7.485458E-28		8.497244E-01	9.297293E-03		9.897894E-01	8.631483E-15		1.000000E+00
29	1.051661E+04	6.563471E-26		8.497244E-01	6.011815E-03		9.958013E-01	5.722286E-15		1.000000E+00
30	1.240291E+04	2.018889E-26		8.497244E-01	3.234168E-03		9.990354E-01	3.116749E-15		1.000000E+00

Using Strain Energy to Identify Areas for Change

Element Strain Energy

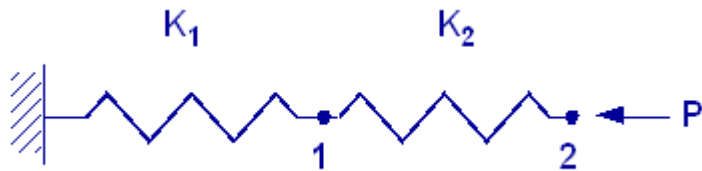
- **Element Strain Energy is an excellent tool for determining which elements to modify the least, but have the greatest potential for impacting results.**
- **In the context of a normal modes solution, the modal element strain energy cannot be compared across modes.**
 - But within a mode, the modal strain energy can still provide useful relative information.
 - If you find the same elements having a large amount of modal strain energy across several modes, these elements are good candidates for modification.
 - Changing the stiffness of the elements with the most ESE in a mode is often the most efficient way to shift the frequency of that mode.
- **In a Rigid Body mode, the strain should be numerical zero, therefore the element strain energy should also be approximately zero.**
- **Element Strain Energy can be defined as**

$$ESE_e = 0.5 * \{ u_e \}^T [K_{ee}] \{ u_e \}.$$

Element Strain Energy (cont.)

- **Example—two springs in series- Which is the best element to stiffen to reduce the tip deflection?**

– Where $K_1 = 10$; $K_2 = 1$; $P = 1$



$$ESE = \frac{1}{2} k \Delta^2$$

$$\Delta_1 = \frac{P}{K_1}$$

$$\Delta_2 = \frac{P}{K_1} + \frac{P}{K_2}$$

$$K_1, ESE_1 = \frac{1}{2} K_1 (\Delta_1)^2 = \frac{1}{2} \frac{(P)^2}{K_1} = \frac{1}{20}$$

$$K_2, ESE_2 = \frac{1}{2} K_2 (\Delta_2 - \Delta_1)^2 = \frac{1}{2} K_2 \left(\frac{P}{K_1} + \frac{P}{K_2} - \frac{P}{K_1} \right)^2$$

$$= \frac{1}{2} K_2 \left(\frac{P}{K_2} \right)^2 = \frac{1}{2}$$

- **Most of the ESE is in the smaller spring. Therefore, stiffening it is the most efficient way to reduce the tip deflection.**

Element Strain Energy (cont.)

- The Case Control Command to trigger the calculation of element strain energy is shown below. These results are available in the f06 file, as well as the binary results file for plotting as fringe results in a post-processor.

ESE Element Strain Energy Output Request

Requests the output of the strain energy in selected elements.

Format:

$$\text{ESE} \left[\left(\begin{array}{c} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right) \left[\begin{array}{c} \text{AVERAGE} \\ \text{AMPLITUDE} \\ \text{PEAK} \end{array} \right] [\text{THRESH} = p] \right) = \left\{ \begin{array}{c} \text{ALL} \\ n \end{array} \right\}$$

Element Strain Energy (cont.)

- **The strain energies are printed**
 - by mode
 - by element type (BAR, QUAD4, etc.),
 - percentage by each element type
 - strain energy density (strain energy / volume, which removes some of the biasing of the element size)
- **For the sample beam model (beam_sol103_10modes.bdf), mode 1 (as well as the other modes) shows the most modal strain energy in element 1**
 - this is where the smallest modification can be made to achieve the biggest effect.

EIGENVALUE = 3.061045E+04				
CYCLES = 2.784549E+01				
E L E M E N T S T R A I N E N E R G I E S				
ELEMENT-TYPE = BEAM		* TOTAL ENERGY OF ALL ELEMENTS IN PROBLEM		= 1.530522E+04
MODE	1	* TOTAL ENERGY OF ALL ELEMENTS IN SET		-1 = 1.530522E+04
ELEMENT-ID	STRAIN-ENERGY	PERCENT OF TOTAL	STRAIN-ENERGY-DENSITY	
1	5.293247E+03	34.5846	8.677454E+05	
2	3.856093E+03	25.1946	6.321463E+05	
3	2.657727E+03	17.3648	4.356929E+05	
4	1.704405E+03	11.1361	2.794107E+05	
5	9.933071E+02	6.4900	1.628372E+05	
6	5.074112E+02	3.3153	8.318217E+04	
7	2.138927E+02	1.3975	3.506438E+04	
8	6.648600E+01	0.4344	1.089934E+04	
9	1.197230E+01	0.0782	1.962672E+03	
10	6.816227E-01	0.0045	1.117414E+02	
TYPE = BEAM	SUBTOTAL	1.530522E+04	100.0000	

Grid Point Kinetic Energy

Grid Point Kinetic Energy

- **Grid Point Kinetic Energy information can be very helpful in identifying how much each DOF is participating in the motion of a particular mode.**
- **Like modal element strain energy, modal grid point kinetic energy cannot be compared across modes.**
 - Within a mode, the participation of DOF in the mode's motion can be seen. If the GPKE is shared across many DOF in the model, this points towards the mode being a global mode, while having a smaller number of DOF showing up with significant GPKE in a mode indicates a local mode.
- **While Element Strain Energy is a stiffness-biased calculation, Grid Point Kinetic Energy is a mass-biased calculation. This can help provide insight into which DOF are moving the most and have the most mass participating in that motion.**

Grid Point Kinetic Energy (cont.)

- **The results of the Grid Point Kinetic Energy request GPKE have the same format as a DISPLACEMENT output.**
 - The values represent the relative amount of kinetic energy in each degree-of-freedom at each GRID point for each mode.
 - The values are scaled such that the total kinetic energy is 1.0 and the printed values are further scaled to be a percentage of the total.
- **For models using the lumped mass formulation, the Grid Point Kinetic energy can be used to examine the distribution of kinetic energy among the grid points within a mode and is computed as:**

$$E_{k_g} = \Phi_g^{mass} \otimes [M_{gg} \Phi_g^{mass}]$$

- **The Grid Point Kinetic Energy output has limited meaning for a coupled mass formulation.**
 - Because the coupled mass formulation produces a coupling of mass across grid points, the sharing of kinetic energy among grid points can occur. In general, this obscures the meaning of the computation.

Grid Point Kinetic Energy (cont.)

- Using the previous cantilevered beam example (sample file beam_sol103_10modes.bdf), looking at the f06 output for mode 1 GPKE, you can see the largest energy is in the T3 direction and pretty well spread around the grids, pointing toward a global mode.

EIGENVALUE = 3.061045E+04								
CYCLES = 2.784549E+01		GRID POINT KINETIC ENERGY (PERCENT)						
		MODE NUMBER = 1						
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
2	G	4.473840E-25	4.475045E-18	1.119432E-02	9.668856E-05	-4.474698E-25	8.307325E-29	
3	G	1.430419E-24	3.435325E-17	1.615365E-01	7.220329E-04	-1.430698E-24	2.632778E-28	
4	G	1.420161E-24	1.105598E-16	7.366569E-01	2.286800E-03	-1.420433E-24	2.590589E-28	
5	G	1.235872E-24	2.476159E-16	2.089146E+00	5.060420E-03	-1.236123E-24	2.234967E-28	
6	G	1.272792E-24	4.522005E-16	4.557218E+00	9.148400E-03	-1.273063E-24	2.283574E-28	
7	G	-8.332398E-26	7.223558E-16	8.408554E+00	1.447745E-02	8.331701E-26	-1.484793E-29	
8	G	-2.084400E-24	1.047493E-15	1.381095E+01	2.079370E-02	2.084720E-24	-3.695156E-28	
9	G	-1.888813E-24	1.409172E-15	2.082973E+01	2.767570E-02	1.889109E-24	-3.336859E-28	
10	G	1.155650E-24	1.782459E-15	2.944991E+01	3.456018E-02	-1.155864E-24	2.038150E-28	
11	G	-1.226244E-25	1.068773E-15	1.980990E+01	2.038739E-02	1.226134E-25	-2.161536E-29	

Grid Point Kinetic Energy (cont.)

- If mode 4 is examined, it can be seen that most of the energy is in the R1 direction, and pretty spread around, indicating a global rotational mode. This agrees with the previous findings using modal effective mass and visualization that mode 4 is a torsional mode.

EIGENVALUE = 6.278638E+06								
CYCLES = 3.987979E+02								
		GRID POINT KINETIC ENERGY (PERCENT)						
		MODE NUMBER = 4						
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
2	G	1.110279E-22	1.218687E-11	-1.320318E-02	6.299606E-01	-4.287709E-23	-6.815079E-23	
3	G	9.040033E-23	7.426815E-11	-1.249530E-01	2.313752E+00	-5.897681E-23	-3.142352E-23	
4	G	2.000574E-22	1.524997E-10	-2.882738E-01	4.589931E+00	-1.863161E-22	-1.374136E-23	
5	G	-5.034499E-23	1.485971E-10	-1.649578E-01	7.029088E+00	7.088074E-23	-2.053578E-23	
6	G	5.670587E-24	3.937376E-12	4.420532E-01	9.450123E+00	-2.864038E-23	2.296980E-23	
7	G	1.642140E-22	-2.036477E-10	1.128654E+00	1.192420E+01	1.439866E-22	-3.082005E-22	
8	G	2.321155E-22	-2.473733E-10	1.210925E+00	1.457582E+01	8.385122E-24	-2.405005E-22	
9	G	6.889136E-23	1.651221E-10	5.033994E-01	1.733278E+01	-1.218186E-23	-5.670961E-23	
10	G	-1.635665E-23	1.271463E-09	-4.962904E-01	1.981656E+01	4.081347E-24	1.227529E-23	
11	G	1.328005E-22	1.594564E-09	-6.010884E-01	1.074152E+01	-3.534794E-23	-9.745258E-23	

Grid Point Kinetic Energy (cont.)

- The Case Control entry for Grid Point Kinetic Energy is:

GPKE

Grid Point Kinetic Energy Output Request

Requests the output of the kinetic energy at selected grid points in normal modes analysis only.

Format:

$$\text{GPKE} \left[\begin{array}{c} \text{PRINT} \\ \text{NOPRINT} \end{array} \right], (\text{PUNCH}, \text{THRESH} = e) = \left\{ \begin{array}{c} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

Describer	Meaning
PRINT	The printer will be the output medium.
NOPRINT	Generates, but does not print, grid point kinetic energy output.
PUNCH	The punch file will be the output medium.
e	Minimum energy threshold. Only energies above this value will be printed and/or punch.
ALL	Grid point kinetic energy for all grid points will be output.
n	Set identification number of a previously appearing SET command. Only grid points with identification numbers that appear on this SET command will be included in output. (Integer > 0)
NONE	Grid point kinetic energy for no points will be output.

Weight Check

WEIGHTCHECK

- **One of the most common errors in normal modes analysis is having incorrect units for the mass properties.**
 - Since the program assumes consistent units, it is up to the user to provide the correct mass properties.
 - From a single DOF system, the frequency is equal the square root of k/m ... therefore, large errors in mass result in large errors in the natural frequency.
 - If the mass density was accidentally entered as weight density, the mass would be artificially large, resulting in modes that are artificially low.
- **As an overall aid in checking out the mass properties in your structure, the Case Control Command WEIGHTCHECK can be used.**
- **The output for the WEIGHTCHECK command (on the g-set level) is similar to that provided by PARAM, GRDPNT (Grid Point Weight Generator, see Appendix B in the Linear Static Analysis User Guide).**

WEIGHTCHECK (cont.)

- The main difference between PARAM,GRDPNT and WEIGHTCHECK is that the weight check may be performed at any or all stages of the mass reduction (see the SET keyword).

WEIGHTCHECK Rigid Body Mass Reduction Check

At each stage of the mass matrix reduction, compute rigid body mass and compare with the rigid body mass t the g-set.

Format:

$$\text{WEIGHTCHECK} \left[\left(\begin{array}{c} \text{PRINT} \\ \text{NOPRINT} \end{array} \right), \text{SET} = \left(\begin{array}{c} \text{G, N, N + AUTOSPC, F, A, V} \\ \text{ALL} \end{array} \right) \right] = \left\{ \begin{array}{c} \text{YES} \\ \text{NO} \end{array} \right\}$$

$$\left[\text{GRID} = \text{gid}, \text{CGI} = \left(\begin{array}{c} \text{YES} \\ \text{NO} \end{array} \right), \left(\begin{array}{c} \text{WEIGHT} \\ \text{MASS} \end{array} \right) \right]$$

- Note:** The units of output (weight or mass) depend on the units of input, it is independent of param,wtmass.
- Using the sample beam model (beam_sol103_10modes.bdf), here are some different set level outputs.

WEIGHTCHECK (cont.)

- Below, you see the printout for the G-Set. This is for all mass in the system after everything has been processed and before any reductions.

```

      O U T P U T   F R O M   W E I G H T   C H E C K
                D E G R E E S   O F   F R E E D O M   S E T   =   G
                R E F E R E N C E   P O I N T   =           0
                        M O
*  4.514000E-05  0.000000E+00  0.000000E+00  0.000000E+00  2.973827E-11 -1.878865E-06 *
*  0.000000E+00  4.514000E-05  0.000000E+00 -2.973827E-11  0.000000E+00  2.257000E-04 *
*  0.000000E+00  0.000000E+00  4.514000E-05  1.878865E-06 -2.257000E-04  0.000000E+00 *
*  0.000000E+00 -2.973827E-11  1.878865E-06  3.423789E-07 -9.394323E-06 -1.486914E-10 *
*  2.973827E-11  0.000000E+00 -2.257000E-04 -9.394323E-06  1.512190E-03 -1.237798E-12 *
* -1.878865E-06  2.257000E-04  0.000000E+00 -1.486914E-10 -1.237798E-12  1.512268E-03 *
                        S
*  1.000000E+00  0.000000E+00  0.000000E+00 *
*  0.000000E+00  1.000000E+00  0.000000E+00 *
*  0.000000E+00  0.000000E+00  1.000000E+00 *
DIRECTION
MASS AXIS SYSTEM (S)      MASS      X-C.G.      Y-C.G.      Z-C.G.
X      4.514000E-05      0.000000E+00  4.162305E-02  6.588009E-07
Y      4.514000E-05      5.000000E+00  0.000000E+00  6.588009E-07
Z      4.514000E-05      5.000000E+00  4.162305E-02  0.000000E+00
                        I (S)
*  2.641748E-07  1.694066E-21  2.584939E-26 *
*  1.694066E-21  3.836900E-04  4.038968E-28 *
*  2.584939E-26  4.038968E-28  3.836900E-04 *
                        I (Q)
*  2.641748E-07 *
*  3.836900E-04 *
*  3.836900E-04 *
                        Q
*  1.000000E+00  0.000000E+00  0.000000E+00 *
*  0.000000E+00  1.000000E+00  0.000000E+00 *
*  0.000000E+00  0.000000E+00  1.000000E+00 *

```

WEIGHTCHECK (cont.)

- Below, you see the printout for the N-Set. Since there are no MPCs in this model, the N-Set is the same as the G-Set.
- Had any MPCs been in the model, it is expected that these results still match the G-Set results- if not, there are errors in the MPCs.
- If it was desired to see the CG and inertia terms, CGI=YES could have been added to the WEIGHTCHECK command.

```

      O U T P U T   F R O M   W E I G H T   C H E C K
      DEGREES OF FREEDOM SET = N
      REFERENCE POINT =      0
      M O
*  4.514000E-05  0.000000E+00  0.000000E+00  0.000000E+00  2.973827E-11 -1.878865E-06 *
*  0.000000E+00  4.514000E-05  0.000000E+00 -2.973827E-11  0.000000E+00  2.257000E-04 *
*  0.000000E+00  0.000000E+00  4.514000E-05  1.878865E-06 -2.257000E-04  0.000000E+00 *
*  0.000000E+00 -2.973827E-11  1.878865E-06  3.423789E-07 -9.394323E-06 -1.486914E-10 *
*  2.973827E-11  0.000000E+00 -2.257000E-04 -9.394323E-06  1.512190E-03 -1.237798E-12 *
* -1.878865E-06  2.257000E-04  0.000000E+00 -1.486914E-10 -1.237798E-12  1.512268E-03 *

      CHANGE WITH RESPECT TO G-SET RIGID BODY MASS/WEIGHT
*      0 %
*
*      0 %
*
*      0 %
*
*      0 %
*
*      0 %
*
*      0 %
*
```

WEIGHTCHECK (cont.)

- Now, the F-Set printout is shown. Here you can see a change with respect to the G-Set.
- As you move down the set processing, the F-Set is the result after any SPCs have been applied.
 - Our cantilevered beam was fixed in all DOF at the left end, so the mass on that grid is removed.

```

      O U T P U T   F R O M   W E I G H T   C H E C K
      DEGREES OF FREEDOM SET = F
      REFERENCE POINT =      0
      M O
*  4.288300E-05  0.000000E+00  0.000000E+00  0.000000E+00  2.825136E-11 -1.784921E-06 *
*  0.000000E+00  4.288300E-05  0.000000E+00 -2.825136E-11  0.000000E+00  2.257000E-04 *
*  0.000000E+00  0.000000E+00  4.288300E-05  1.784921E-06 -2.257000E-04  0.000000E+00 *
*  0.000000E+00 -2.825136E-11  1.784921E-06  3.252599E-07 -9.394323E-06 -1.486914E-10 *
*  2.825136E-11  0.000000E+00 -2.257000E-04 -9.394323E-06  1.512190E-03 -1.175908E-12 *
* -1.784921E-06  2.257000E-04  0.000000E+00 -1.486914E-10 -1.175908E-12  1.512264E-03 *

      CHANGE WITH RESPECT TO G-SET RIGID BODY MASS/WEIGHT
*      -5 %
*
*      -5 %
*
*      -5 %
*
*      -5 %
*
*      0 %
*
*      0 %

```

WEIGHTCHECK (cont.)

- After the F-Set, we see the A-Set.
- Since there were no further reductions in the sample cantilevered beam (no ASET or OSET definitions), the A-Set is no different from the F-Set.

```

      O U T P U T   F R O M   W E I G H T   C H E C K
      DEGREES OF FREEDOM SET = A
      REFERENCE POINT =      0
      M O
*  4.288300E-05  0.000000E+00  0.000000E+00  0.000000E+00  2.825136E-11 -1.784921E-06 *
*  0.000000E+00  4.288300E-05  0.000000E+00 -2.825136E-11  0.000000E+00  2.257000E-04 *
*  0.000000E+00  0.000000E+00  4.288300E-05  1.784921E-06 -2.257000E-04  0.000000E+00 *
*  0.000000E+00 -2.825136E-11  1.784921E-06  3.252599E-07 -9.394323E-06 -1.486914E-10 *
*  2.825136E-11  0.000000E+00 -2.257000E-04 -9.394323E-06  1.512190E-03 -1.175908E-12 *
* -1.784921E-06  2.257000E-04  0.000000E+00 -1.486914E-10 -1.175908E-12  1.512264E-03 *

      CHANGE WITH RESPECT TO G-SET RIGID BODY MASS/WEIGHT
*      -5 % *
*      -5 % *
*      -5 % *
*      -5 % *
*      0 % *
*      0 % *
```

Mode Selection

Mode Selection

- **When executing a modal response analysis, it is possible that not all of the computed modes are required for a good dynamic solution.**
- **Often, only the lowest few modes are needed.**
 - It is quite common to evaluate the frequency content of transient loads and determine a frequency above which no modes are expected to be excited.
 - The act of specifically not using all of the modes of a system in the solution is termed mode truncation.
 - Mode truncation assumes that an accurate solution can be obtained using a reduced set of modes.
 - The number of modes used in a solution is controlled in a modal transient response analysis through a number of methods.

Mode Selection (cont.)

- **Methods for defining the included modes**
 - The frequency range selected on the eigenvalue entry (EIGRL or EIGR) is one means to control the frequency.
 - Three parameters are available to limit the number of modes included in the solution.
 - PARAM,LFREQ gives the lower limit on the frequency range of retained modes
 - PARAM,HFREQ gives the upper limit on the frequency range of retained modes
 - PARAM,LMODES gives the number of the lowest modes to be retained.
 - These parameters can be used to include the desired set of modes. Note that the default is for all computed modes to be retained.
- **The MODESELECT Case Control command permits the user to specify ALL data related to mode selection without the need for any parameters.**

Mode Selection (cont.)

- **The MODESELECT command, which can be employed for selecting either structure modes or fluid modes, offers five different and distinct options.**
 1. Mode selection based on arbitrary mode numbers.
 2. Mode selection based on the number of lowest modes. This option is similar to the usage of the LMODES/LMODESFL parameter.
 3. Mode selection based on range of mode numbers.
 4. Mode selection based on frequency range. This option is similar to the usage of the LFREQ/LFREQFL and HFREQ/HFREQFL parameters. However, this option is more general since it also allows for the UNCONDITIONAL inclusion or exclusion of selected modes regardless of their frequencies.
 5. Mode selection based on modal effective mass fraction (MEFFMFRA) criteria. This powerful option allows the user to select modes based on different MEFFMFRA criteria. Further, like Option (4) above, it also allows for the UNCONDITIONAL inclusion or exclusion of selected modes regardless of their MEFFMFRA values.

Mode Selection (cont.)

- **Remember that truncating modes in a particular frequency range may truncate a significant portion of the behavior in that frequency range.**
 - Typically, high-frequency modes are truncated because they are more costly to compute.
 - Truncating high-frequency modes truncates high frequency response.
 - You should evaluate the truncation in terms of the loading frequency and the important characteristic frequencies of the structure.
- **Methods for improving solution accuracy when truncating modes are discussed in Section 5 (Residual Vectors).**
 - The ‘correction’ is done by appending additional shapes to the retained mode set to account for the quasi-static influence of the truncated high-frequency modes.

Modal Contribution Factor

Modal Contribution Factor

- **The response of large structural components to loads in the time domain can be determined using either a direct or modal solution approach.**
 - The modal approach is typically employed for large models when the response of the structure can be approximated using only a small number of its normal modes.
 - During this process, data recovery and post-processing of large quantities of response data usually occurs.
 - Often, the user desires detailed information for several points of interest in the model.
- **One item of particular interest is often referred to as the *modal contribution fraction* or the *modal participation factor*. As these names imply, the value for this item is defined as that portion of the total response that can be attributed to a particular normal mode.**

Modal Contribution Factor (cont.)

- **Modal analysis solutions are based on the theory that a small number of normal modes can be used to represent the response of the structure.**
- **The eigenvectors of these modes are used to transform the problem from the physical domain into a scalar domain of generalized coordinates.**
 - This transformation results in a much smaller problem size.
 - The solution back to the physical domain is obtained by transformation, again using the eigenvectors of the natural modes. The response at a physical degree of freedom (a grid point's translation or rotation component) for a particular time step can be found by summing the individual modal responses.
 - The calculation of modal contribution fractions can be performed as part of the total response calculation.

Modal Contribution Factor (cont.)

- The MCFRACTION Case Control command is designed to give the user control over the amount and ordering of the produced modal contribution fraction data.
- The general format of the command is:

MCFRACTION

Modal Contribution Fractions Output Request

Requests the form and type of modal contribution fraction output.

Format:

$$\text{MCFRACTION} \left(\left[\text{STRUCTURE} \right], \left[\begin{array}{c} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right] \left[\begin{array}{c} \text{REAL or IMAG} \\ \text{PHASE} \end{array} \right], \left[(\text{SORT} = \text{sorttype}) \right], \right.$$

$$\left. \left[\begin{array}{c} \text{KEY} = \text{FRACTION} \\ \text{sortitem} \end{array} \right], \left[\begin{array}{c} \text{ITEMS} = \text{FRACTION} \\ \text{ALL} \\ (\text{itemlist}) \end{array} \right] \left[\begin{array}{c} \text{SOLUTION} = \text{ALL} \\ \text{m} \\ \text{NONE} \end{array} \right] \left[\begin{array}{c} \text{FILTER} = 0.001 \\ \text{fratio} \end{array} \right]$$

$$\left. \left[\text{NULL} = \begin{array}{c} 12 \\ \text{ipowr} \end{array} \right] \right) = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}$$

Modal Contribution Factor (cont.)

- The SET Case Control command has been enhanced to recognize and process a new format for use with the MCFRACTION Case Control command.
- This new format allows the analyst to specify a set of grid point and component code combinations.
 - An example of the new format is “SET 1000 = 917/T3, 85/R2, 127016/T1” which demonstrates the general input format of grid ID and component code separated by a slash.
 - No THRU ranges are permitted.

Modal Contribution Factor (cont.)

- **The MCFRACTION command is available in modal frequency response (SOL 111), modal transient response (SOL 112) and modal complex eigenvalue analysis (SOL 110) solution sequences only.**
- **If superelements are used, its use is restricted to residual structure data recovery operations only.**
- **The modal contribution fraction computation is intended for a small sub-set of the degrees of freedom present in the model.**
 - The calculations are limited by the amount of memory available to store the physical response recovery matrix. Its columns represent a response for each mode and its rows each represent one of the grid point components for which a physical solution is requested.
 - This matrix must fit in the available memory.
- **This command can result in a lot of output in the f06 file, use it sparingly**

Workshop 1

Model Checks

- **Workshop Objectives**
 - Edit a sample file and perform various model checks
 - Review the f06 file to become familiar with the output

SECTION 4

Direct Matrix Input

Direct Matrix Input

- In addition to generating the mass, stiffness, and damping matrix automatically from the FE model, you can add mass, stiffness, and damping to specific DOFs.
- In dynamics modeling, we frequently need to include special nonfinite element effects such as mechanical devices, servomechanisms, smart structures, and matrices from external structures. These effects can usually be included as extra terms in the system matrices.
- This is done using a **DMIG** entry
 - 2 Types of DMIGs described on next 2 slides
- **Some examples of when a DMIG might be used:**
 - User supplied section of the model that doesn't have an FEM representation
 - Superelements (the reduced matrices can be stored as DMIGs)
 - Test (the test-developed stiffness can be represented as matrices stored as DMIGs)

Direct Matrix Input

- **Type 1: G-Type Matrices**

- G-type matrices are the size of the G-SET.
- G-type matrices are applied at the system level prior to any reduction or constraint operations (M-SET, S-SET, etc).
- G-type matrices are real and symmetric. They are selected in the Case Control Section via:
 - M2GG = name of mass matrix addition
 - B2GG = name of damping matrix addition
 - K2GG = name of stiffness matrix addition
- G-type matrices can be added to superelements and the residual structure.

Direct Matrix Input (cont.)

- **Type 2: P-Type Matrices**

- P-type matrices are the same size as the P-set (G-SET plus E-SET for extra points).
- P-type matrices are not applied at the system level prior to constraint operations.
- The P-type DMIGs are processed through the constraints and reduction procedures in parallel with the G-type matrices, then added to the reduced model (A- or H-SET) before the analysis operation.
- Note that modal reduction (modal solutions) operations do not include effects for P-type matrices.
- Because added at the end, P-type DMIGs don't have to be symmetric.
- The load reduction operations to the analysis sets do not include the effects of P-type matrices.
- For some applications, the P-type DMIG is the only way to add terms to the equation of motion.
 - An example is a brake squeal analysis, where the friction terms added are skew-symmetric.

Direct Matrix Input

- **The g-type DMIGs are added to the model up front in the G-SET, prior to any SET reduction (G-SET to M-SET, N-SET to F-SET, etc)**
 - It is required to be symmetric
 - generally the recommended approach
- **The p-type DMIGs are added after all the reductions (including modal reduction).**
 - Because it is added at the end, p-type DMIGs don't have to be symmetric
 - For some applications, the p-type DMIG is the only way to add terms to the equation of motion.
 - An example is a brake squeal analysis, where the friction terms added are skew-symmetric.

Direct Matrix Input (cont.)

- **P-type matrices need not be real or symmetric. They are selected in the Case Control Section via:**
 - M2PP = name of mass matrix addition
 - B2PP = name of damping matrix addition
 - K2PP = name of stiffness matrix addition
- **P-type matrices can only be added to the residual structure. They cannot be added to superelements.**
 - PARAM, WTMASS does not affect direct input matrices M2GG or M2PP. PARAM, CM2 can be used to scale M2GG.

DMIG Bulk Data Entry Description

DMIG

Direct Matrix Input at Points

Defines direct input matrices related to grid, extra, and/or scalar points. The matrix is defined by a single header entry and one or more column entries. A column entry is required for each column with nonzero elements.

Header Entry Format:

1	2	3	4	5	6	7	8	9	10
DMIG	NAME	"0"	IFO	TIN	TOUT	POLAR		NCOL	

Column Entry Format:

DMIG	NAME	GJ	CJ		G1	C1	A1	B1	
	G2	C2	A2	B2	-etc.-				

Example:

DMIG	STIF	0	1	3	4				
DMIG	STIF	27	1		2	3	3.+5	3.+3	
	2	4	2.5+10	0	50		1	0	

Field	Contents
NAME	Name of the matrix. See Remark 1. (One to eight alphanumeric characters, the first of which is alphabetic.)
IFO	Form of matrix input. IFO = 6 must be specified for matrices selected by the K2GG, M2GG, and B2GG Case Control commands. (Integer) 1 = Square 9 or 2 = Rectangular 6 = Symmetric
TIN	Type of matrix being input: (Integer) 1 = Real, single precision (One field is used per element.) 2 = Real, double precision (One field is used per element.) 3 = Complex, single precision (Two fields are used per element.) 4 = Complex, double precision (Two fields are used per element.)
TOUT	Type of matrix that will be created: (Integer) 0 = Set by precision system cell (Default) 1 = Real, single precision 2 = Real, double precision 3 = Complex, single precision 4 = Complex, double precision

DMIG Bulk Data Entry Description (cont.)

Field	Contents
POLAR	Input format of Ai, Bi. (Integer=blank or 0 indicates real, imaginary format; Integer > 0 indicates amplitude, phase format.)
NCOL	Number of columns in a rectangular matrix. Used only for IFO = 9. (Integer > 0)
GJ	Grid, scalar or extra point identification number for column index. (Integer > 0)
CJ	Component number for grid point GJ. (0 < Integer ≤ 6; blank or zero if GJ is a scalar or extra point.)
Gi	Grid, scalar, or extra point identification number for row index. (Integer > 0)
Ci	Component number for Gi for a grid point. (0 < CJ ≤ 6; blank or zero if Gi is a scalar or extra point.)
Ai, Bi	Real and imaginary (or amplitude and phase) parts of a matrix element. If the matrix is real (TIN = 1 or 2), then Bi must be blank. (Real)

Remarks:

1. Matrices defined on this entry may be used in dynamics by selection in the Case Control with K2PP = NAME, B2PP = NAME, M2PP = NAME for $[K_{pp}]$, $[B_{pp}]$, or $[M_{pp}]$, respectively. Matrices may also be selected for all solution sequences by K2GG = NAME, B2GG = NAME, and M2GG = NAME. The g-set matrices are added to the structural matrices before constraints are applied, while p-set matrices are added in dynamics after constraints are applied. Load matrices may be selected by P2G = NAME for dynamic and superelement analyses.
2. The header entry containing IFO, TIN and TOUT is required. Each nonnull column is started with a GJ, CJ pair. The entries for each row of that column follows. Only nonzero terms need be entered. The terms may be input in arbitrary order. A GJ, CJ pair
3. Field 3 of the header entry must contain an integer 0.

DMIG Bulk Data Entry Description (cont.)

4. For symmetric matrices (IFO = 6), a given off-diagonal element may be input either below or above the diagonal. While upper and lower triangle terms may be mixed, a fatal message will be issued if an element is input both below and above the diagonal.
5. The recommended format for rectangular matrices requires the use of NCOL and IFO = 9. The number of columns in the matrix is NCOL. (The number of rows in all DMIG matrices is always either p-set or g-set size, depending on the context.) The GJ term i
6. If NCOL is not used for rectangular matrices, two different conventions are available:
 - If IFO = 9, GJ and CJ will determine the sorted sequence, but will otherwise be ignored; a rectangular matrix will be generated with the columns submitted being in the 1 to N positions, where N is the number of logical entries submitted (not counting th
 - If IFO = 2, the number of columns of the rectangular matrix will be equal to the index of the highest numbered non-null column (in internal sort). Trailing null columns of the g- or p-size matrix will be truncated.
7. The matrix names must be unique among all DMIGs.
8. TIN should be set consistent with the number of decimal digits required to read the input data adequately. For a single-precision specification on a short-word machine, the input will be truncated after about eight decimal digits, even when more digits are needed, a double precision specification should be used instead. However, note that a double precision specification requires a "D" type exponent even for terms that do not need an exponent. For example, unity may be input as 1.0 in single precision, but the longer form 1.0D0 is required for double precision.
9. On long-word machines, almost all matrix calculations are performed in single precision and on short-word machines, in double precision. It is recommended that DMIG matrices also follow these conventions for a balance of efficiency and reliability. The recommended value for TOUT is 0, which instructs the program to inspect the system cell that measures the machine precision at run time and sets the precision of the matrix to the same value. TOUT = 0 allows the same DMIG input to be used on any machine. If TOUT is contrary to the machine type specified (for example, a TOUT of 1 on a short-word machine), unreliable results may occur.
10. If any DMIG entry is changed or added on restart then a complete re-analysis is performed. Therefore, DMIG entry changes or additions are not recommended on restart.

Scale Factor for X2GG and X2PP Matrices

- The K2GG, M2GG, B2GG, and P2G commands accept the specifications of multiple matrices and real scale factor(s)

Example:

K2GG = 1.25*KDMIG

M2GG = 2.8*MSCMIG1, 3.9*MSCMIG2

- K2PP, M2PP, and B2PP commands accept the specifications of multiple matrices and complex or real scale factor(s)

Example:

K2PP = KDMIG

K2PP = KDMIG1, KDMIG2, KDMIG3

K2PP = 2.04*KDMIG1, 0.82*KDMIG2

K2PP = (2.04, .5)*KDMIG1, (0.82, 0.)*KDMIG2

Direct Input of Structural Element Damping Matrices

- Can also add structural element damping by using the K42GG command and DMIG entries
- Can also specify multiple matrices

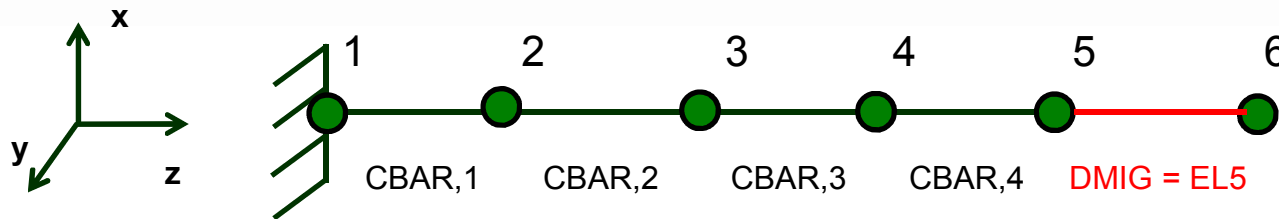
Example:

K42GG = KDMIG

K42GG = KDMIG1, KDMIG2, KDMIG3

K42GG = 2.04*KDMIG1, 0.82*KDMIG2

DMIG Example



The stiffness and mass for the connection from Grid 5 to Grid 6 is being supplied directly... this model only has 2 DOF per Grid- DOF 3 (TZ) and DOF 5 (RY).

$$K_5 = \begin{matrix} & \begin{matrix} 5,3 & 5,5 & 6,3 & 6,5 \end{matrix} \\ \begin{matrix} 5,3 \\ 5,5 \\ 6,3 \\ 6,5 \end{matrix} & \left[\begin{array}{cccc} 500039. & & & \\ -250019. & 166680. & & \\ -500039. & 250019. & 500039. & \\ -250019. & 83340. & 250019. & 166680. \end{array} \right] \end{matrix}$$

SYMMETRIC

$$M_5 = \begin{matrix} & \begin{matrix} 5,3 & 5,5 & 6,3 & 6,5 \end{matrix} \\ \begin{matrix} 5,3 \\ 5,5 \\ 6,3 \\ 6,5 \end{matrix} & \left[\begin{array}{cccc} 3.5829 & 0. & 0. & 0. \\ 0. & 0. & 0. & 0. \\ 0. & 0. & 3.5829 & 0. \\ 0. & 0. & 0. & 0. \end{array} \right] \end{matrix}$$

Input File for the DMIG Example

```
$ DMIG EXAMPLE
SOL 103 $ NORMAL MODES ANALYSIS
CEND
TITLE = DMIG TO READ STIFFNESS AND MASS FOR ELEM 5
SUBTITLE = PLANAR PROBLEM
SPC = 10
$ SPECIFY K2GG AND M2GG
K2GG = EXSTIF
M2GG = EXMASS
METHOD = 10
$
BEGIN BULK
$
EIGRL, 10, , , 2
$
CBAR,1,1,1,2,10
CBAR,2,1,2,3,10
CBAR,3,1,3,4,10
CBAR,4,1,4,5,10
$
PBAR,1,1,2.654-3,5.869-7
$
MAT1,1,7.1+10,,.33 2700.
$
GRID,1,,0.,0.,0.,,1246
GRID,2,,1.,0.,0.,,1246
GRID,3,,2.,0.,0.,,1246
GRID,4,,3.,0.,0.,,1246
GRID,5,,4.,0.,0.,,1246
GRID,6,,5.,0.,0.,,1246
GRID,10,,0.,0.,10.,,123456
$
```

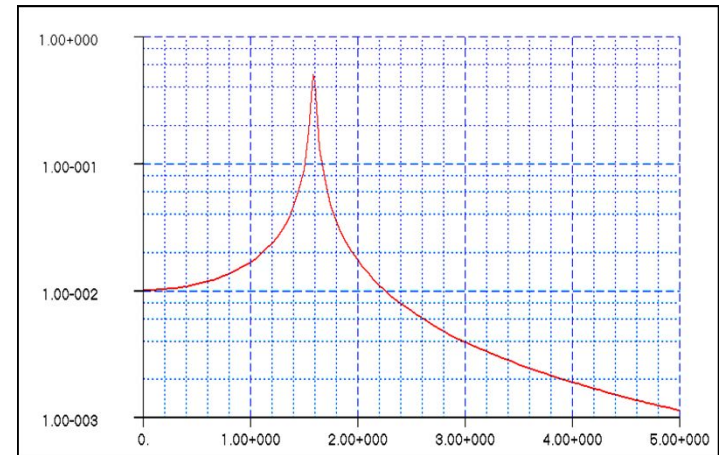
```
$
$ HEADER ENTRY FOR STIFFNESS
DMIG,EXSTIF,0,6,1
$
$ DATA ENTRIES FOR STIFFNESS
DMIG,EXSTIF,5,3,,5,3,500039.
+,5,5,-250019.,,6,3,-500039.
+,6,5,-250019.
$
DMIG,EXSTIF,5,5,,5,5,166680.
+,6,3,250019.,,6,5,83340.
$
DMIG,EXSTIF,6,3,,6,3,500039.
+,6,5,250019.
$
DMIG,EXSTIF,6,5,,6,5,166680.
$
$ HEADER ENTRY FOR MASS
DMIG,EXMASS,0,6,1
$
$ DATA ENTRIES FOR MASS
DMIG,EXMASS,5,3,,5,3,3.5829
$
DMIG,EXMASS,6,3,,6,3,3.5829
$
SPC1,10,123456,1
$
ENDDATA
```

SECTION 5

Residual Vectors

Concept of Modal Approach

- For many problems the dynamic solution involves only the lower normal modes, not those above a threshold value of frequency. The modes above the threshold only respond statically.
- For these cases, only a relatively small subset of all normal modes need be computed. The normal modes above the threshold can be ignored.
- The specification of the subset of retained normal modes is termed modal truncation.



Remember how a SDOF system responds as the excitation frequency changes:

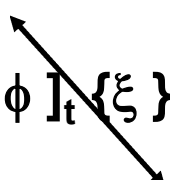
- at forcing frequencies much below the natural frequency, the dynamic response is static
- at forcing frequencies much higher than the natural frequency, the dynamic response approaches zero.

Concept of Modal Approach (cont.)

- Assume the response can be represented as a linear combination of the calculated modes: $\{ U \} = [\phi] \{ \xi \}$
- Number of possible modes = number of degrees of freedom with mass on them
- Same results as direct if all modes are retained
 - Not practical
 - Defeats purpose of modal approach

Concept of Modal Approach (cont.)

- Partition the modes into retained and truncated modes

$$\{ \mathbf{U} \} = [\boldsymbol{\phi}] \{ \boldsymbol{\xi} \} = [\boldsymbol{\phi}]_r \{ \boldsymbol{\xi} \}_r + [\boldsymbol{\phi}]_t \{ \boldsymbol{\xi} \}_t$$


where $[\boldsymbol{\phi}]_r$ = modes that are retained

$[\boldsymbol{\phi}]_t$ = truncated modes

- $[\boldsymbol{\phi}]_r$ is usually a small subset of $[\boldsymbol{\phi}]$
- Quality of the modal solution depends on how well a linear combination of the retained modes, $[\boldsymbol{\phi}]_r$, can represent the actual solution due to the applied loads.

How Do I Compensate for the Missing Modal Content?

- **Two methods are available to compensate for the missing modal content**
 - Residual Vectors (Recommended method and is turned on by default for most modal solutions)
 - Mode Acceleration (not used any longer)

Residual Vector Method

- Augment modes with static vectors obtained from various sources (detailed later)
- The response of the neglected modes tends to be static if the truncated frequencies are high compared to the excitation frequency
 - As $(\omega/\omega_n) \ll 1$, dynamic magnification $\longrightarrow 1$
 - Excellent approximation of missing modes if the above condition is satisfied
- Improves modal solutions in all cases
- Recommended to be included for all response analyses using the modal approach (the default)

Residual Vector Method (cont.)

- **Supports superelement analysis**
- **Two eigenvalue tables are printed**
 - Original eigenvalue table
 - Second eigenvalue table with the additional eigenvalues appended at the end - one for each additional residual vector
- **Re-orthogonalization is performed to ensure that linearly dependent vectors are removed**

Sources For Residual Vectors

- **Residual Vectors are simply static displacements from various sources that are added to make sure that a linear combination of the modes can represent the static solution. Residual Vectors come from the following sources.**
 - Applied Loads in SOLs 111 & 112 – triggered by RLOAD & TLOAD Excite ID entries (not triggered by Subcase DLOAD callout)
 - Applied Loads in SOL 103 – also triggered by Static Subcase / Load entries, in addition to any RLOAD / TLOADs existing in deck
 - Inertia – $6 \text{ Mass} \times \text{unit rigid body acceleration loads}$, for constrained & superelement components
 - Damping – triggered by CDAMP, CVISC, & damping CBUSH elements – to give accurate damping load thru element, that existing K & M based modes Residual Vectors cannot provide
 - RVDOFs – Unit DOF load Residual Vectors, an alternative to applied loads w/o using RLOAD or TLOAD data. Also useful for internal superelement response DOF.
 - Initial Conditions

Residual Vector Processing

- **All residual vector loads are concatenated together, then filtered to ensure linear independence, then calculate shapes based on linearly independent loads, and re-orthogonalize with existing modes**
 - Ensure loads are linearly independent from modal inertial forces
 - Determine base vectors from static response due to loads
 - Ensure base vectors are linearly independent
 - Independent from modal vectors
 - Independent from other base vectors
 - Orthogonalize base vectors with respect to the modal vectors to produce residual vectors. These vectors will result in diagonal mass and stiffness matrices.

Residual Vector Case Control Entry

RESVEC

Controls Residual Vectors

Specifies options for and calculation of residual vectors.

Format:

$$\text{RESVEC} \left[\left[\begin{array}{c} \text{INRLOD} \\ \text{NOINRL} \end{array} \right], \left[\begin{array}{c} \text{APPLOD} \\ \text{NOAPPL} \end{array} \right], \left[\begin{array}{c} \text{ADJLOD} \\ \text{NOADJLOD} \end{array} \right], \left[\begin{array}{c} \text{RVDOF} \\ \text{NORVDO} \end{array} \right], \left[\begin{array}{c} \text{DAMPLOD} \\ \text{NODAMP} \end{array} \right], \left[\begin{array}{c} \text{DYNRSP} \\ \text{NODYNRSP} \end{array} \right] \right] =$$
$$\left\{ \begin{array}{c} \text{SYSTEM/NOSYSTEM} \\ \text{COMPONENT/NOCOMPONENT} \\ \text{BOTH or YES} \\ \text{NO} \end{array} \right\}$$

Examples:

```
RESVEC=SYSTEM  
RESVEC (NOINRL) =COMPONENT  
RESVEC=NO
```

Residual Vector Case Control Entry (cont.)

Describer	Meaning
INRLD/ NOINRL	Controls calculation of residual vectors based on inertia relief (Default = INRLD).
APPLD/ NOAPPL	Controls calculation of residual vectors based on applied loads (Default = APPLD).
ADJLOD/ NOADJLOD	Controls calculation of residual vectors based on adjoint load vectors (SOL 200 only; Default = ADJLOD).
RVDOF/ NORVDOF	Controls calculation of residual vectors based on RVDOFi entries (Default = RVDOF).
DAMPLD/ NODAMP	Controls calculation of residual vectors based on viscous damping (Default = DAMPLD).
DYNRSP/ NODYNRSP	Controls whether the residual vectors will be allowed to respond dynamically in the modal transient or frequency response solution. See Remark 5. (Default = DYNRSP).
SYSTEM/ NOSYSTEM	Controls calculation of residual vectors for system (a-set) modes. For NOSYSTEM, describers inside the parentheses are ignored. See Remark 2. for default.
COMPONENT/ NOCOMPONENT	Controls calculation of residual vectors for component (superelement or o-set) modes. For NOCOMPONENT, describers inside the parentheses are ignored. See Remark 2. for default.
BOTH or YES	Requests calculation of residual vectors for both system modes and component modes. See Remark 2. for default.
NO	Turns off calculation of residual vectors for both system and component modes, and describers inside the parentheses are ignored. See Remark 2. for default.

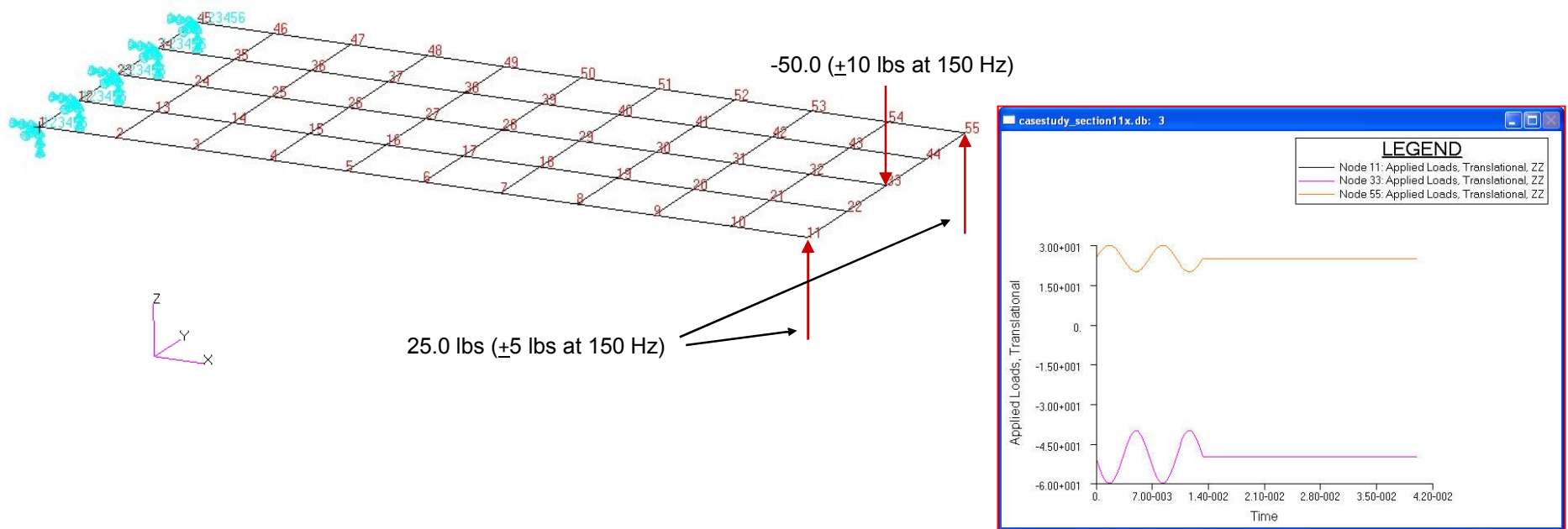
Residual Vector Case Control Entry (cont.)

Remarks:

1. RESVEC=SYSTEM/NOSYSTEM and RESVEC=COMPONENT/NOCOMPONENT may be specified in the same subcase.
2. RESVEC=BOTH is the default in all solution sequences except SOLs 103, 106, (with PARAM,NMLOOP), and 115, wherein RESVEC=COMPONENT is the default.
3. If the RESVEC command is specified then the user parameters PARAM,RESVEC and PARAM,RESVINER are ignored.
4. The lower frequency cutoff on the EIGR or EIGRL Bulk Data entries should be left blank or set to a value below the minimum frequency. Residual vectors may not be calculated if all modes below the maximum frequency cutoff are not determined. If low frequency modes are to be excluded from the analysis, use the MODESELECT Case Control command or PARAM,LFREQ.
5. Caution needs to be exercised when allowing the residual vectors to respond dynamically in a modal solution. The best approach is to always include enough normal modes to capture the dynamics of the problem, and rely on the residual vectors to help account for the influence of the truncated modes on the quasistatic portion of the response. This is not the default setting for this capability. When choosing to allow the residual vectors to respond dynamically, it is important to be aware of the frequency content of the excitation, as it will have the ability to excite these augmentation modes. If this is undesirable, then the forcing function should be filtered in advance to remove any undesired frequency content, or specify the NODYNRSP keyword.

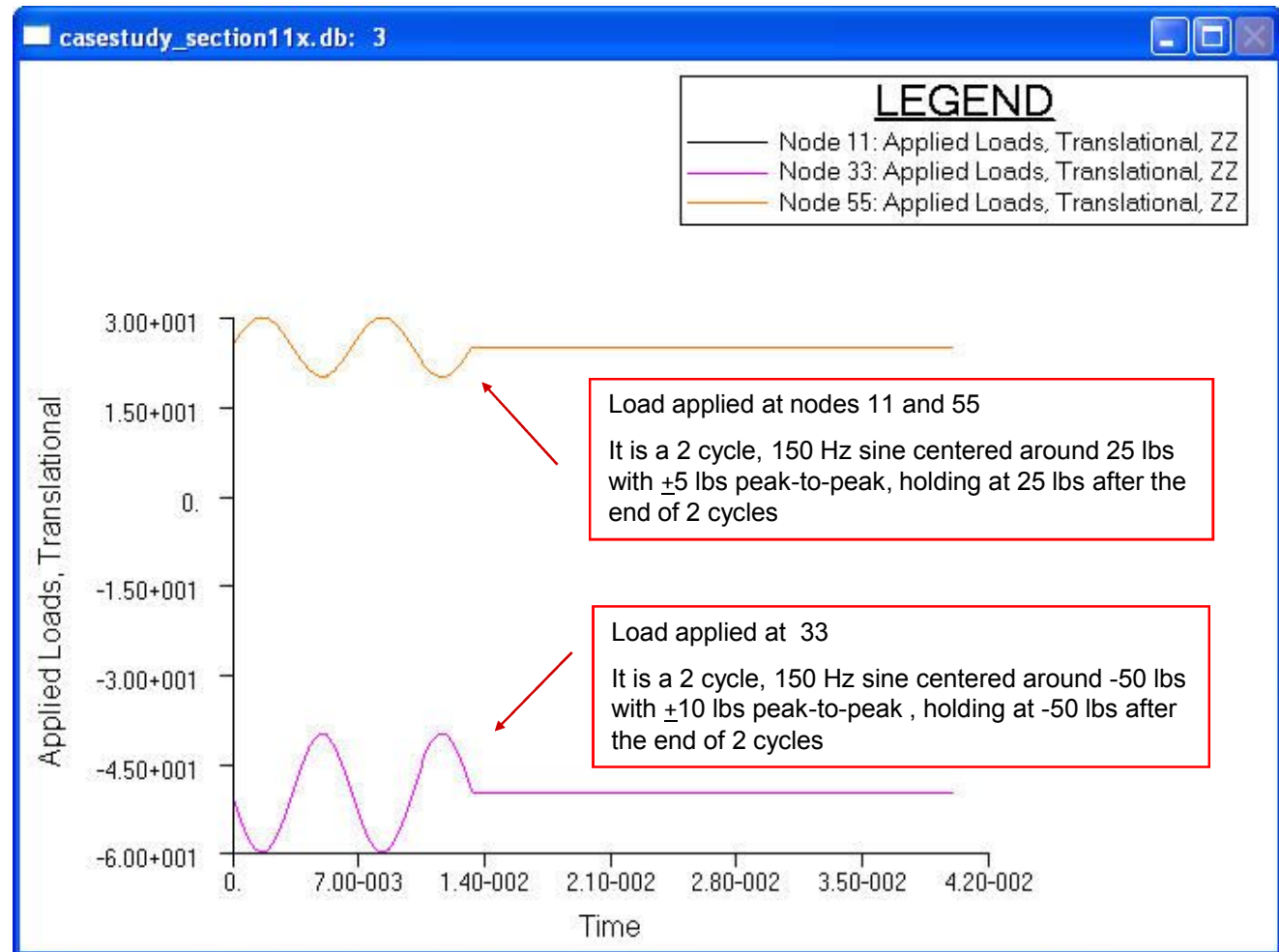
Case Study: Residual Vectors

- Apply a two cycle sine load at three nodes as seen in the picture below. The model is a 5" x 2" plate. The thickness of the plate is 0.1". The load is at 150 Hz, and in the Z-direction. The material properties are $E = 30 \times 10^6$ psi, $\nu = 0.3$, and $\rho = 0.282 \times 0.00259$.



Case Study: Residual Vectors (cont.)

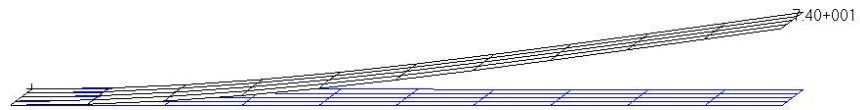
- Applied load



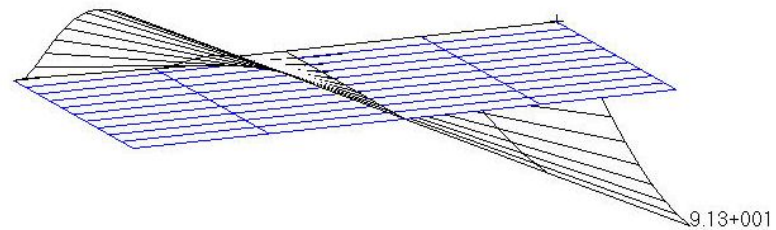
Case Study: Residual Vectors (cont.)

- **First 3 mode shapes**

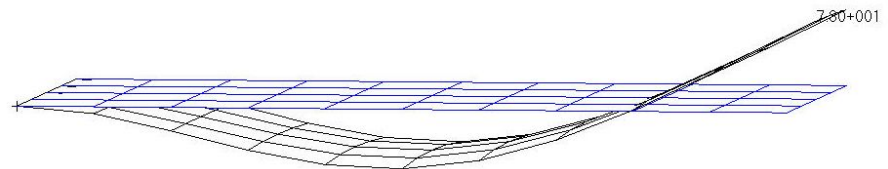
Mode 1- 133.17 Hz



Mode 2- 648.72 Hz



Mode 3- 821.38 Hz



Case Study: Residual Vectors (cont.)

- **Transient analyses to be performed**
 - Modal analysis *WITHOUT* RESVEC
 - Use RESVEC=NO in the Case Control section
 - Use just the first 3 modes (133.17, 648.72, 821.38 Hz)
 - Use modal damping of 3% of critical
 - Modal analysis *WITH* RESVEC
 - In the Case Control section, set RESVEC=YES (this is the default for most modal solution sequences, e.g. SOL 112, modal transient response)
 - Use just the first 3 modes (133.17, 648.72, 821.38 Hz)
 - Use modal damping of 3% of critical
 - Direct analysis
 - The effect is similar to including all modes, this will be our baseline
 - Damping of g (G) = 0.06, and w_3 ($W3$) = 836.7 rad/sec (133.17 Hz... the mode nearest and in the direction of our 150 Hz excitation frequency)
 - Note that $g=0.06$ is close to, but not exactly equal to, 3% critical damping

Case Study: Residual Vectors (cont.)

- **Specify time step and damping data for modal analysis**
 - Integration step size: highest mode of interest=821.38 Hz...
period=.001217 sec... at least 10 integration pts per cycle= .0001217...
round to .0001 sec
 - Duration: at least 3 cycles of lowest mode of interest (133.17 Hz) *past* end of applied force (2 cycles at 150 Hz=0.01333 sec);
 $0.01333 + (1/133.17) * 3 = 0.03586...$ round to 0.04 sec
 - Number of steps = duration/step size= $0.04/0.0001=400$
 - As usual, check response for verification

Case Study: Residual Vectors (cont.)

- .bdf file for modal transient analysis, NO RESVEC

```
$ Linear Transient Response Analysis, Modal Formulation
```

```
SOL 112
```

```
$
```

```
CEND
```

```
TITLE = no resvec, no ic
```

```
ECHO = NONE
```

```
RESVEC = NO
```

```
$
```

```
SUBCASE 1
```

```
METHOD = 1
```

```
TSTEP = 1
```

```
SPC = 2
```

```
DLOAD = 2
```

```
DISPLACEMENT (PLOT, SORT2, REAL) = ALL
```

```
ACCELERATION (PLOT, SORT2, REAL) = ALL
```

```
SPCFORCES (PLOT, SORT2, REAL) = ALL
```

```
OLOAD (PLOT, SORT2, REAL) = ALL
```

```
STRESS (PLOT, SORT2, REAL, VONMISES, BILIN) = ALL
```

```
$
```

```
BEGIN BULK
```

```
PARAM POST 0
```

```
PARAM PRTMAXIM YES
```

```
TABDMP1 1 CRIT
```

```
0. .03 10000. .03 ENDT
```

```
BEGIN BULK
```

```
PARAM POST 0
```

```
PARAM PRTMAXIM YES
```

```
TSTEP 1 400 1.-4
```

```
EIGRL 1 1000. 0
```

```
$ Loads for Load Case : applied_load
```

```
SPCADD 2 1
```

```
TLOAD1 6 3 1
```

```
TLOAD1 8 4 1
```

```
TLOAD1 10 5 1
```

```
DLOAD 2 1. 1. 6 1. 8 1. 10
```

```
$ Displacement Constraints of Load Set : fixed
```

```
SPC1 1 123456 1 12 23 34 45
```

```
$ Nodal Forces of Load Set : force
```

```
FORCE 3 55 0 1. 0. 0. 1.
```

```
FORCE 4 11 0 1. 0. 0. 1.
```

```
FORCE 5 33 0 2. 0. 0. -1.
```

```
$ Dynamic Load Table : 150hz_2cycles
```

```
TABLED1 1
```

```
* -.004 25. 0. 25.
```

```
.
```

```
.04 25. .044 25. ENDT
```

```
ENDDATA
```

Case Study: Residual Vectors (cont.)

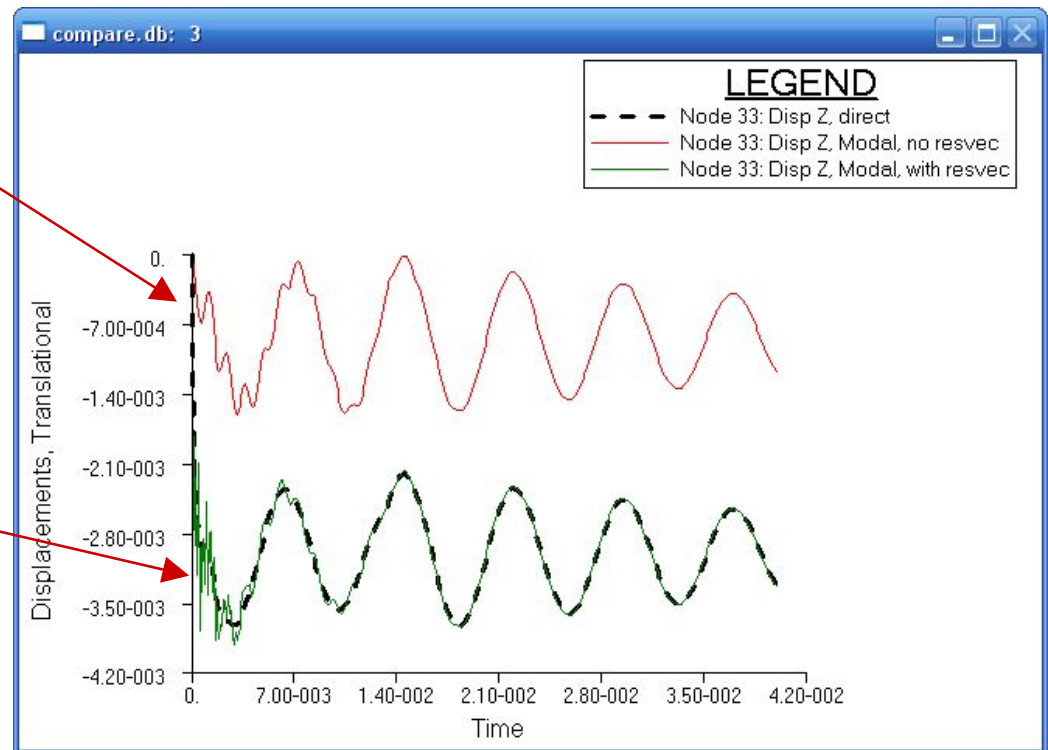
- .f06 eigenvalue table showing before/after augmentation

REAL EIGENVALUES (BEFORE AUGMENTATION OF RESIDUAL VECTORS)						
MODE NO.	EXTRACTION ORDER	EIGENVALUE	RADIANS	CYCLES	GENERALIZED MASS	GENERALIZED STIFFNESS
1	1	7.001032E+05	8.367217E+02	1.331684E+02	1.000000E+00	7.001032E+05
2	2	1.661386E+07	4.076010E+03	6.487171E+02	1.000000E+00	1.661386E+07
3	3	2.663469E+07	5.160880E+03	8.213796E+02	1.000000E+00	2.663469E+07

REAL EIGENVALUES (AFTER AUGMENTATION OF RESIDUAL VECTORS)						
MODE NO.	EXTRACTION ORDER	EIGENVALUE	RADIANS	CYCLES	GENERALIZED MASS	GENERALIZED STIFFNESS
1	1	7.001031E+05	8.367216E+02	1.331684E+02	1.000000E+00	7.001031E+05
2	2	1.661386E+07	4.076010E+03	6.487171E+02	1.000000E+00	1.661386E+07
3	3	2.663469E+07	5.160880E+03	8.213796E+02	1.000000E+00	2.663469E+07
4	4	1.754894E+08	1.324724E+04	2.108364E+03	1.000000E+00	1.754894E+08
5	5	2.060401E+08	1.435410E+04	2.284525E+03	1.000000E+00	2.060401E+08
6	6	2.194601E+08	1.481419E+04	2.357751E+03	1.000000E+00	2.194601E+08
7	7	6.550708E+08	2.559435E+04	4.073468E+03	1.000000E+00	6.550708E+08
8	8	8.879831E+08	2.979904E+04	4.742665E+03	1.000000E+00	8.879831E+08
9	9	1.413599E+09	3.759786E+04	5.983886E+03	1.000000E+00	1.413599E+09
10	10	2.697120E+09	5.193380E+04	8.265521E+03	1.000000E+00	2.697120E+09
11	11	4.135841E+09	6.431050E+04	1.023533E+04	1.000000E+00	4.135841E+09
12	12	4.250384E+09	6.519497E+04	1.037610E+04	1.000000E+00	4.250384E+09

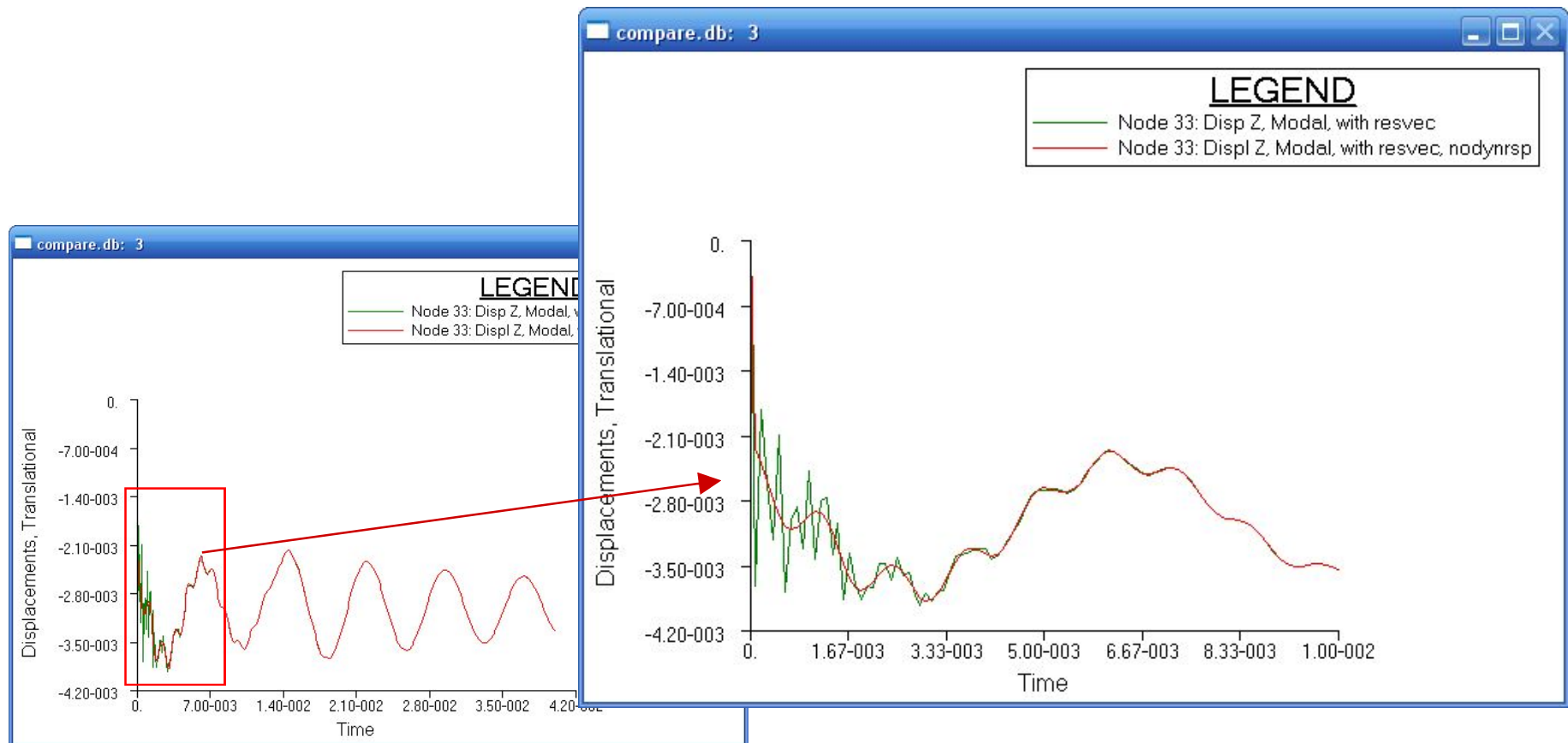
Case Study: Residual Vectors (cont.)

- Comparison for the displacement response at node 33, z-direction between Direct, no RESVEC, and with RESVEC
- Notice the lost 'quasi-static' response without the Residual Vectors
- Notice the high-freq response at the beginning... this is due to the non-zero initial force being seen as a 'step' input to the model
- The next slide shows how the use of the NODYNRSP flag helps the high-frequency response



Case Study: Residual Vectors (cont.)

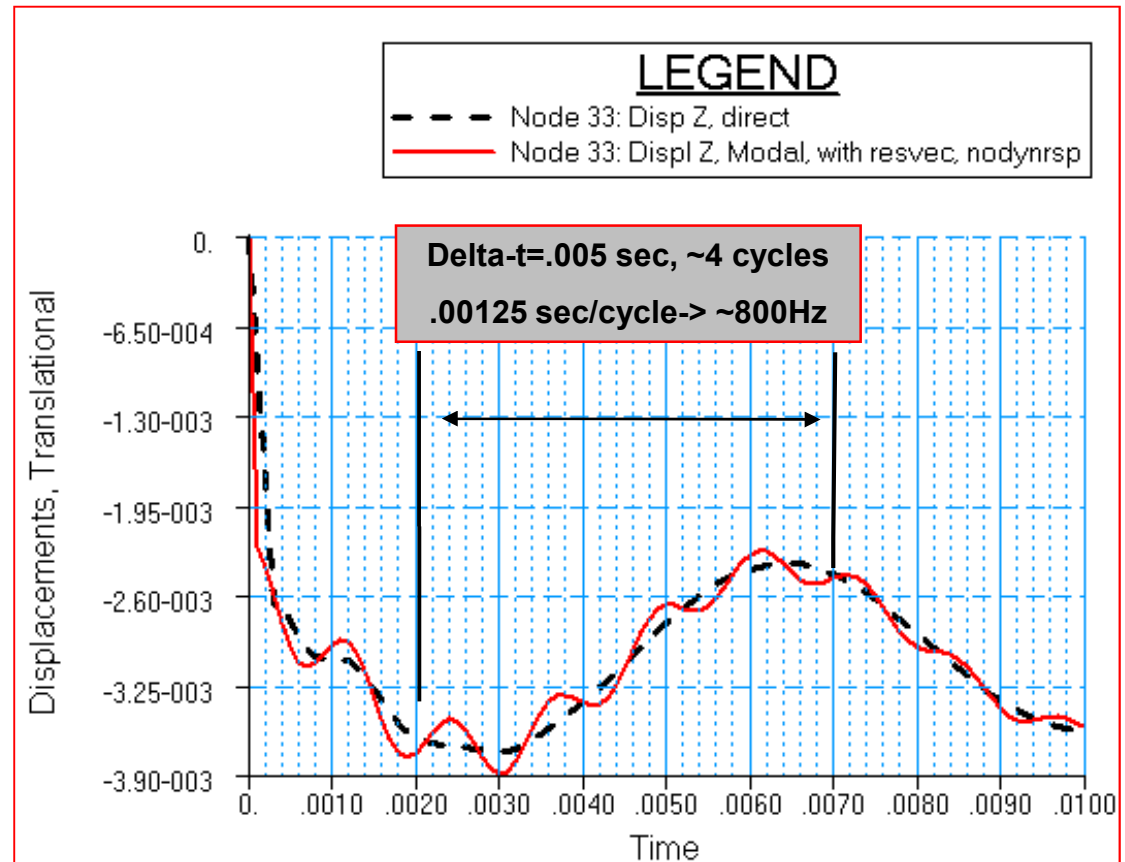
- Comparison for the displacement response at node 33, z-direction between RESVEC and RESVEC with NODYNRSP



Case Study: Residual Vectors (cont.)

- **Comparison for the displacement response at node 33, z-direction between Direct and RESVEC with NODYNRSP**

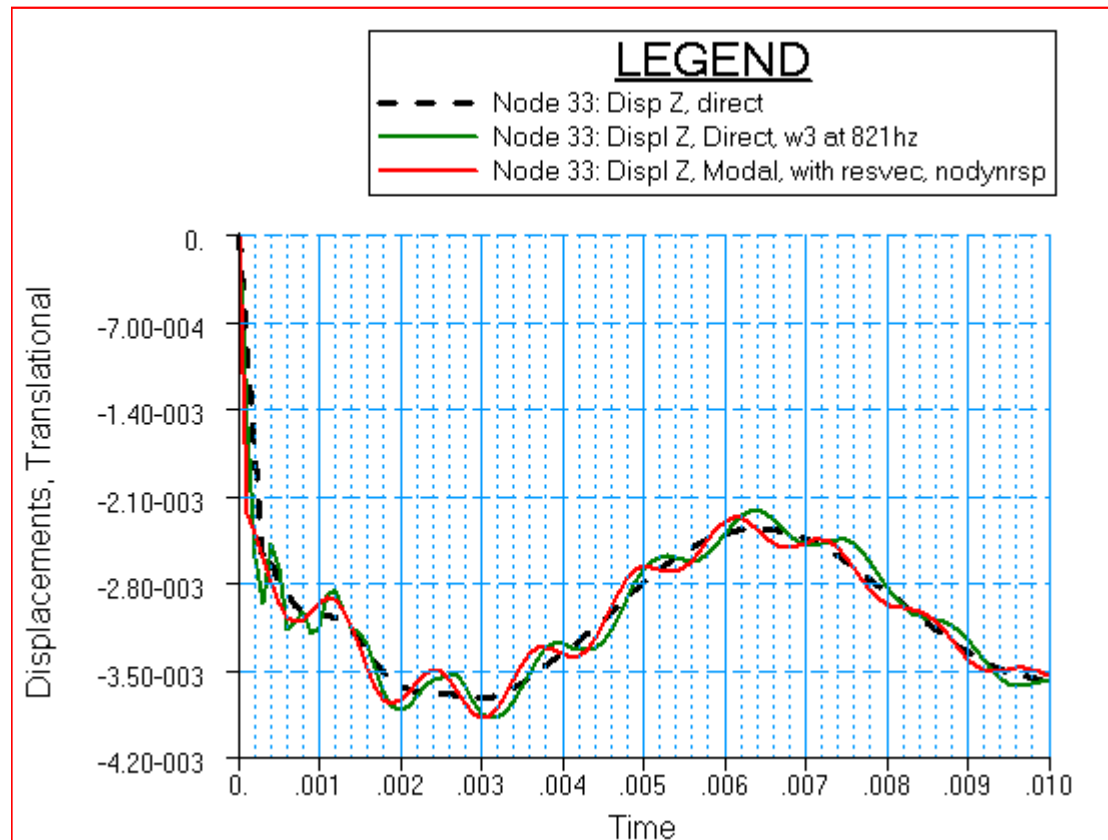
- Compared to the Direct solution, RESVEC NODYNRSP is still showing oscillations ...why?
- The answer is the step input at the start excites ALL modes, not just the residual vectors... the oscillation you see is actually the 821 Hz normal mode responding... the 648 Hz mode isn't as likely to be excited by the direction of this loading.
- So, why doesn't the direct solution show the 821 Hz oscillation? See the answer on the next slide...



Case Study: Residual Vectors (cont.)

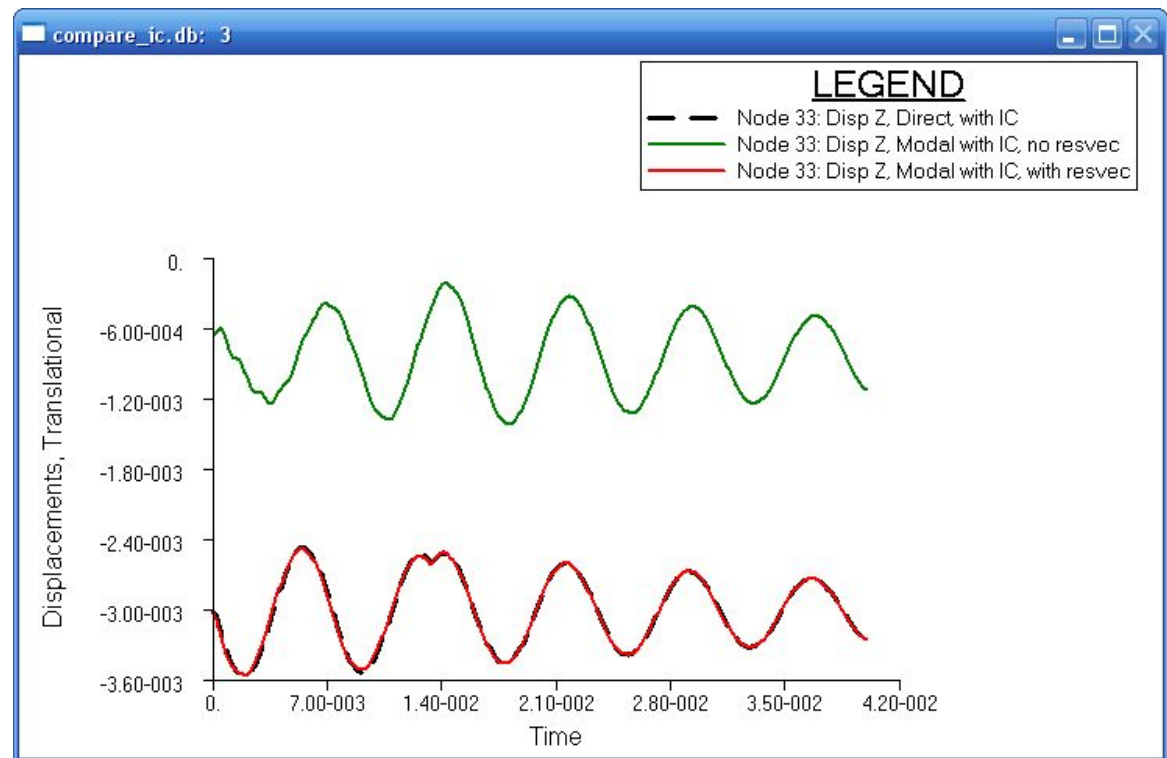
- **Comparison between Direct with W3 at 133 Hz and W3 at 821**

- When setting up the Direct solution, structural damping of 6% was used... since a transient response analysis can not use structural damping, a conversion to viscous damping was done, and the conversion frequency chosen was 133 Hz (to match the mode expected to contribute the most to the response)
- Due to the damping 'error' at other frequencies, the 821 Hz mode received too much damping... as can be seen in the graph, if the conversion was done at 821 Hz, the oscillation exists



Case Study: Residual Vectors (cont.)

- **Comparison for the displacement response at node 33, z-direction between Direct and RESVEC with NODYNRSP**
 - When using Initial Conditions, the effect of the 'step' or non-zero starting force is better handled, such that NODYNRSP is less necessary
 - This would be problem dependent... there can be other 'triggers' that might excite high frequencies such as noise in the forcing function or other near-steps in the loading, so the use of NODYNRSP is recommended



Case Study: Residual Vectors (cont.)

- **Summary**

- As can be seen in the plots, using Residual Vectors greatly improved the solution, bringing it very close to the Direct solution
 - Note that $g=.06$ is close to but not exactly equivalent to 3% critical damping
 - Also, the conversion to viscous damping is only 'valid' at the conversion frequency
- Furthermore, the flag NODYNRSP helped remove any dynamic contribution of the Residual Vector response
- The use of Initial Conditions is important when there is a non-zero initial loading in the forcing function... an alternate approach would be to slowly ramp the loading to the correct initial state

Residual Vectors: Closing Comments

- In a problem that has quasi-static response, it is important that Residual Vectors be used
- Residual Vectors can be considered a ‘safety net’ to help mitigate issues of modal truncation
- If the displaced shape of the loading applied statically cannot be represented by a linear combination of the retained modes, then the answers will not be correct... regardless of the excitation frequency
- It is important to capture the spatial aspect of the loading in addition to the frequency
- Be careful of unfiltered or step-like forcing functions (use NODYNRSP)

SECTION 6

Complex Eigenvalue Analysis

Complex Eigenvalue Analysis

- **Assess the stability of systems modeled with transfer functions (including servomechanisms and rotating systems)**
- **Computes damped modes of systems**
- **Mass and stiffness matrices may be unsymmetric and may contain complex numbers.**
- **For additional details on Complex Eigenvalue Analysis, see the *MSC.Nastran Dynamics User's Guide*.**

Applications of Complex Eigenvalue Analysis

- **Some applications of complex eigenvalue**
 - Damped frequencies
 - For lightly damped systems, complex roots are very close to real roots
 - Rotordynamics
 - Flutter Analysis
 - Brake Squeal Analysis
 - Coriolis mass flowmeter
 - etc.

Complex Eigensolutions - Theory

- Equation of motion

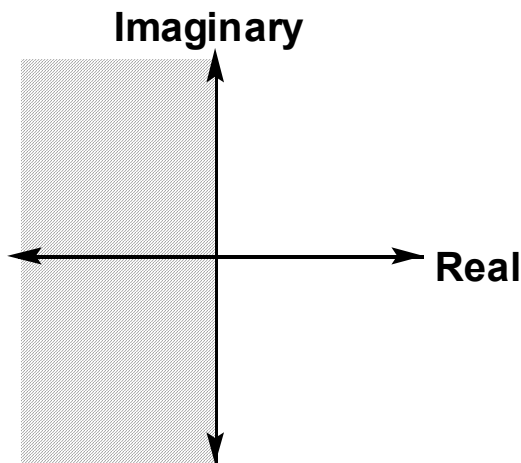
$$[Mp^2 + Bp + K]\{u\} = \{0\}$$

where $p = \alpha + i\omega$

and $\alpha = \text{real part of solution}$

$\omega = \text{imaginary part of solution}$

- For stable systems, $\alpha < 0$



- Damping coefficient

$$g \approx -2\alpha / |\omega| = 2\xi$$

Complex Eigensolutions in MSC Nastran

- **B matrix is the same as that used in frequency response analysis.**
- **The direct method solves the equation with M,B,K matrices of D-size (physical variables plus extra points).**
- **Modal solves the equation with M,B,K matrices of H-size (modal coordinates plus extra points). Undamped modes are first computed to transform matrices from D-size to H-size.**
- **Three methods of complex eigenanalysis: HESS, CLAN, IRAM (plus two obsolete methods: INV, DET)**
- **HESS (upper Hessenberg) is related to GIV. This method requires nonsingular M and may be quite expensive for large problems. Therefore, the modal method is usually recommended except for small problems.**

Complex Eigensolutions in MSC Nastran (cont.)

- **HESS: solves canonical equation forms.**

- Transformation method
- There are two categories:
 - Systems with $[B] = 0$
 - Systems with $[B] \neq 0$

- **$[B] = 0$ case solves**

$$[[\mathbf{A}] - \lambda[\mathbf{I}]]\{\phi\} = 0 \qquad [\mathbf{A}] = -[\mathbf{M}^{-1}\mathbf{K}], \lambda = p^2$$

- **$[B] \neq 0$ case studies:** $[[\mathbf{A}] - p[\mathbf{I}]]\{\phi\} = 0$

$$\text{where } \{\phi\} = \begin{Bmatrix} \mathbf{u} \\ \mathbf{v} \end{Bmatrix}$$

$$[\mathbf{A}] = \begin{bmatrix} 0 & \mathbf{I} \\ -\mathbf{M}^{-1}\mathbf{K} & -\mathbf{M}^{-1}\mathbf{B} \end{bmatrix}$$

Complex Eigenvalue Method

HESS Method

Format:

1	2	3	4	5	6	7	8	9	10
EIGC	SID	METHOD	NORM	G	C	E	ND0		

- Typically only METHOD and ND0 are needed
- Example

EIGC	99	HESS					4		

Complex Eigenvalue Method (cont.)

CLAN Method

- CLAN - Similar to real Lanczos - hybrid of a transformation and a tracking method.
- **Many cases only METHOD and ND0 are needed**

Format:

1	2	3	4	5	6	7	8	9	10
EIGC	SID	METHOD	NORM	G	C	E	ND0		

- **Example**

EIGC	99	CLAN					4		

Complex Eigenvalue Method (cont.)

CLAN Method (cont.)

- There are cases if you suspect that you are missing roots, you can specify continuation entries—one for each search region

Format:

1	2	3	4	5	6	7	8	9	10
EIGC	SID	METHOD	NORM	G	C	E	ND0		
	ALPHA AJ	OMEGA AJ	ALPHA BJ	OMEGA BJ	LJ		NDJ		

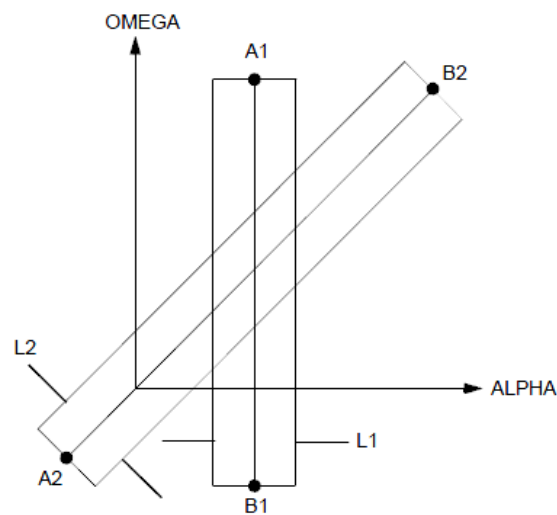
- ALPHA AJ, OMEGA AJ – end point A_j of jth search region (default=0.0)
- ALPHA BJ, OMEGA BJ – end point B_j of jth search region (default=0.0)
- LJ – width for the jth search region (default=1.0)
- NDJ – desired number of roots and eigenvectors desired at the jth search region
- One continuation for each additional search region

Complex Eigenvalue Method (cont.)

CLAN Method (cont.)

- **Example**

EIGC	99	CLAN					20		
	-1.5	-1.5	5.0	5.0	1.2		4		
	2.0	5.0	2.0	-1.4	1.2		6		



Complex Eigenvalue Method (cont.)

CLAN Method (cont.)

- Typically you specify around the imaginary axis
- Use the real eigenvalue results as a starting point

Complex Eigenvalue Method (cont.)

IRAM Method

- IRAM (Implicitly Restarted Arnoldi Method)
- Krylov subspace method
- Based on the ARPACK public domain code

<http://www.caam.rice.edu/software/ARPACK/>

- **Mass matrix must be symmetric**
- **Left eigenvectors are not computed for IRAM method**
- **May not be suitable for SOL 200**

Complex Eigenvalue Method (cont.)

- **General Performance**

- Much faster than HESS
- Similar performance as CLAN for small problem
- Can be significantly slower than CLAN for large problem (e.g., brake squeal)

Format and Example

EIGC	SID	Method					ND		
EIGC	10	IRAM					4		

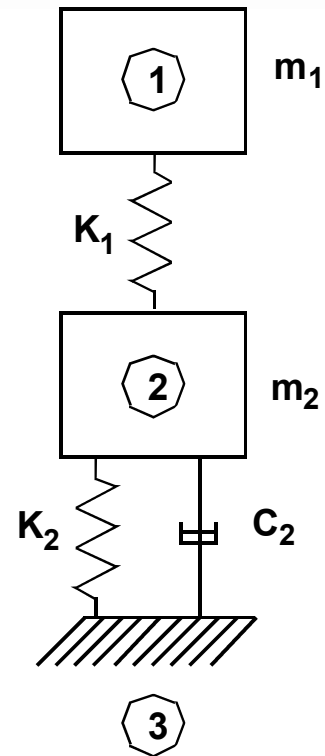
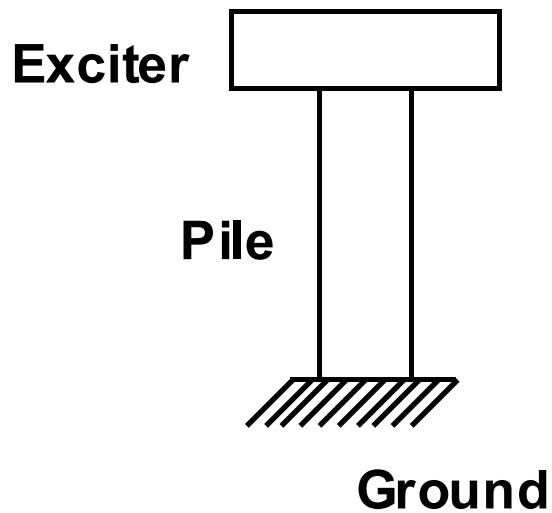
Input Required for Complex Eigenanalysis

- **Executive Control Section**
SOL (for required input see below)

	Structured Solution
Direct	107
Modal	110

- **Case Control Section**
CMETHOD (required for both)
METHOD (required for modal)
- **Bulk Data Section**
EIGC (required for both)
EIGR or EIGRL (required for modal)

Example 1 – Complex Eigenvalue



m_1	3.0 lb-sec ² /in
m_2	1.5 lb-sec ² /in
K_1	50,000 lb/in
K_2	12,500 lb/in
C_2	30 lb-sec/in

Input Listing for Example 1

```
$ sec06_example1.dat
$
SOL 107
CEND
TITLE= TWO-DOF MODEL (IMAC 8, PG 891)
SUBTITLE= COMPLEX MODES
DISPLACEMENT= ALL $ DEFAULT= REAL, IMAGINARY
SPC= 100
CMETHOD= 99
$
BEGIN BULK
$
$ COMPLEX EIGENVALUE EXTRACTION PARAMETERS
$
EIGC, 99, HESS, , , , 4
$
$ DEFINE GRIDS, MASSES, AND STIFFNESSES
$ GRID 1 = EXCITER (X=2, MASS=3) 50K STIFFNESS BETWEEN GRIDS 1 AND 2
$ GRID 2 = PILE (X=1, MASS=3) 12.5K STIFFNESS BETWEEN GRIDS 2 AND 3
$ GRID 3 = BASE (X=0, FIX BASE)
$
GRID, 1, , 2., 0., 0.
GRID, 2, , 1., 0., 0.
GRID, 3, , 0., 0., 0.
GRDSET, , , , , , 23456
CELAS2, 1, 50000., 1, 1, 2, 1
CELAS2, 2, 12500., 2, 1, 3, 1
CONM2, 201, 1, , 3.0
CONM2, 202, 2, , 1.5
SPC, 100, 3, 1
$
$ DEFINE DAMPER OF 30 BETWEEN GRIDS 2 AND 3
$
CVISC, 101, 1, 2, 3
PVISC, 1, 30.
$
ENDDATA
```

Output for Example 1

COMPLEX EIGENVALUE SUMMARY									
0	ROOT NO.	EXTRACTION ORDER	EIGENVALUE		FREQUENCY		DAMPING		
			(REAL)	(IMAG)	(CYCLES)		COEFFICIENT		
	1	1	-2.660969E+00	-4.983521E+01	7.931520E+00		1.067907E-01		
	2	2	-2.660969E+00	4.983521E+01	7.931520E+00		1.067907E-01		
	3	3	-7.339031E+00	-2.360312E+02	3.756553E+01		6.218695E-02		
	4	4	-7.339031E+00	2.360312E+02	3.756553E+01		6.218695E-02		
1	TWO-DOF MODEL (IMAC 8, PG 891)					DECEMBER 21, 2011	MSC.NASTRAN 11/25/11	PAGE	11
0	COMPLEX MODES								
	COMPLEX EIGENVALUE = -2.660969E+00, -4.983521E+01								
	COMPLEX EIGENVECTOR NO. 1								
	(REAL/IMAGINARY)								
	POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
0	1	G	1.000000E+00	0.0	0.0	0.0	0.0	0.0	
			0.0	0.0	0.0	0.0	0.0	0.0	
0	2	G	8.514119E-01	0.0	0.0	0.0	0.0	0.0	
			1.591320E-02	0.0	0.0	0.0	0.0	0.0	
0	3	G	0.0	0.0	0.0	0.0	0.0	0.0	
			0.0	0.0	0.0	0.0	0.0	0.0	
1	TWO-DOF MODEL (IMAC 8, PG 891)					DECEMBER 21, 2011	MSC.NASTRAN 11/25/11	PAGE	12
0	COMPLEX MODES								
	COMPLEX EIGENVALUE = -2.660969E+00, 4.983521E+01								
	COMPLEX EIGENVECTOR NO. 2								
	(REAL/IMAGINARY)								
	POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
0	1	G	-1.000000E+00	0.0	0.0	0.0	0.0	0.0	
			0.0	0.0	0.0	0.0	0.0	0.0	
0	2	G	-8.514119E-01	0.0	0.0	0.0	0.0	0.0	
			1.591320E-02	0.0	0.0	0.0	0.0	0.0	
0	3	G	0.0	0.0	0.0	0.0	0.0	0.0	
			0.0	0.0	0.0	0.0	0.0	0.0	

Output for Example 1 (cont.)

```

1 TWO-DOF MODEL (IMAC 8, PG 891) DECEMBER 21, 2011 MSC.NASTRAN 11/25/11 PAGE 13
  COMPLEX MODES
0
  COMPLEX EIGENVALUE = -7.339031E+00, -2.360312E+02
                        C O M P L E X   E I G E N V E C T O R   N O .           3
                        (REAL/IMAGINARY)

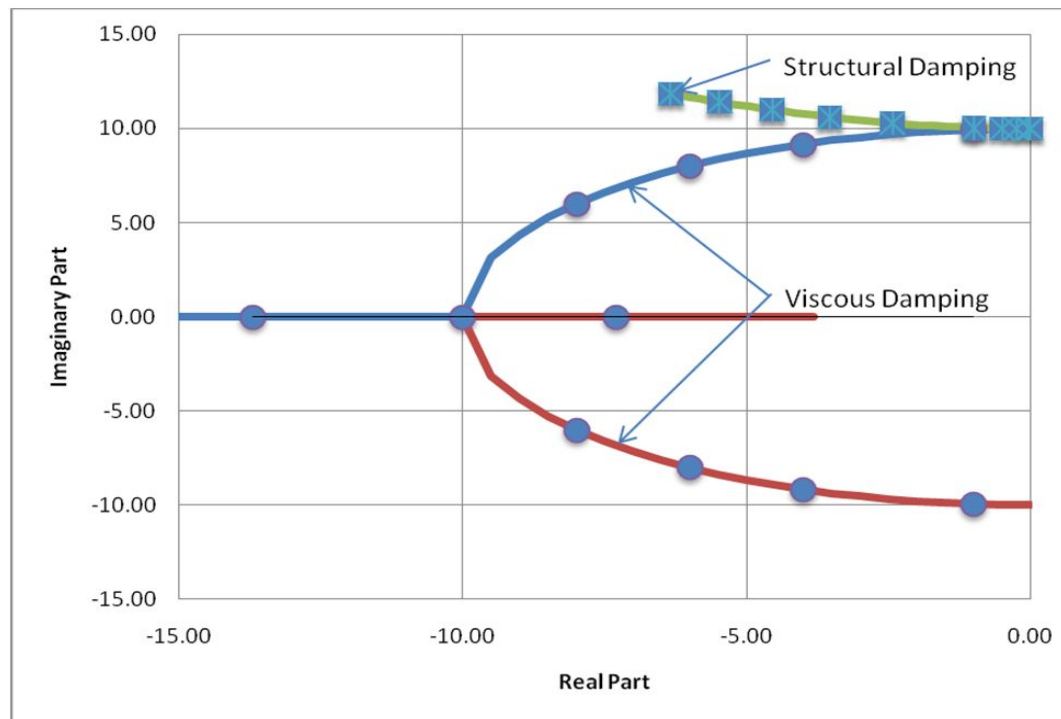
  POINT ID.  TYPE      T1      T2      T3      R1      R2      R3
0           1      G    -4.241094E-01  0.0      0.0      0.0      0.0      0.0
0           2      G    -3.768431E-02  0.0      0.0      0.0      0.0      0.0
0           3      G     1.000000E+00  0.0      0.0      0.0      0.0      0.0
0           3      G     0.0      0.0      0.0      0.0      0.0      0.0
0           3      G     0.0      0.0      0.0      0.0      0.0      0.0
1 TWO-DOF MODEL (IMAC 8, PG 891) DECEMBER 21, 2011 MSC.NASTRAN 11/25/11 PAGE 14
  COMPLEX MODES
0
  COMPLEX EIGENVALUE = -7.339031E+00,  2.360312E+02
                        C O M P L E X   E I G E N V E C T O R   N O .           4
                        (REAL/IMAGINARY)

  POINT ID.  TYPE      T1      T2      T3      R1      R2      R3
0           1      G     4.241094E-01  0.0      0.0      0.0      0.0      0.0
0           2      G    -3.768431E-02  0.0      0.0      0.0      0.0      0.0
0           3      G    -1.000000E+00  0.0      0.0      0.0      0.0      0.0
0           3      G     0.0      0.0      0.0      0.0      0.0      0.0
0           3      G     0.0      0.0      0.0      0.0      0.0      0.0
0           3      G     0.0      0.0      0.0      0.0      0.0      0.0

```

Damping effects on Complex Eigenvalue

- **Viscous damping typically (not always) decreases the frequency as compared to the undamped frequency**
 - Complex conjugate pairs
- **Structural damping typically increases the frequency as compared to the undamped frequency**



General Recommendations

- **HESS method solves for all complex eigenvalues**
 - Can be used for small problem
 - For large problem, use the modal approach
- **Complex Lanczos is more efficient for large problem**
 - Unlike real eigenvalue, you may miss roots if the search regions are not properly defined
 - No equivalent of Sturm sequence check
- **Always use modal approach for large problem**
- **For lightly damped system, you can perform a real eigenvalue analysis to get a good estimate of the frequencies**

Workshop 2

- **Workshop Objectives**
 - Learn how to set up a complex eigenvalue problem using
 - HESS method
 - CLAN method
 - Learn to perform both the direct and modal methods

SECTION 7

EXTRA POINTS, TRANSFER FUNCTIONS, AND NOLINS

Extra Points

- **As discussed in Section 4, some DMIG input are not symmetric. In that case, one cannot define these terms using a grid entry.**
- **An extra point (EPOINT) can be used for this purpose.**
 - added to the end after the reduction (added to the D-SET)
 - For a direct method, the A-SET and E-SET constitutes the D-SET. For the modal method, the EPOINTS, H-SET, and D-SET are related by the following relationship.

$$\{\mathbf{u}_d\} = [\phi_{dh}]\{\mathbf{u}_h\}$$

$$[\phi_{dh}] = \begin{bmatrix} \phi_{ai} & 0 \\ 0 & \mathbf{I}_{ee} \end{bmatrix}$$

$$\{\mathbf{u}_h\} = \begin{bmatrix} \xi_i \\ \mathbf{u}_e \end{bmatrix}$$

Extra Points (Cont.)

- Nonstructural DOFs utilized to represent nonstructural variables.
- Defined by EPOINT Bulk Data entry
- Exist only in the residual structure for the dynamics solutions (E-set, subset of D-set)
- Unaffected by any reduction procedure including modal reduction
- Cannot be used as structural DOFs
- Cannot be constrained by MPCs or SPCs
- Can only be used in P-type direct input matrices or transfer functions
- Can only undergo dynamic loading with the use of the DAREA Bulk Data entry

Transfer Functions

- **Transfer functions are used to define dynamic functions of the form:**

$$(b_0 + b_1 p + b_2 p^2)u_d + \sum_{i=1}^N (a_0 + a_1 p + a_2 p^2)_i u_i = 0$$

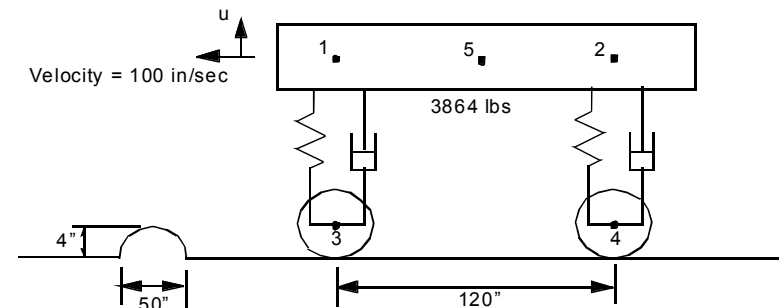
where u_d = dependent coordinate
 u_i = independent degree of freedom
 p = differential operator ($p = d/dt$)

- **Equivalent to P-type DMIG matrices (M2PP, B2PP, K2PP)**
- **Function is added to other P-type matrix input. u_d defines the row with b_0 , b_1 , and b_2 specifying the diagonal terms. u_i defines the column for the a_0 , a_1 , and a_2 terms.**
- **Defined by the TF Bulk Data entry and selected by the TFL Case Control command (residual structure only)**

Transfer Function Example

- For example, in Workshop 3, a transfer function is used to keep track of the relative motion between a wheel and chassis.
- EPOINTS are used as a placeholder for the result of the transfer function
 - For one of the tires:

```
$ DEFINE EXTRA POINTS TO HOLD DIFFERENCES
$ BETWEEN WHEELS AND CARRIAGE
$
EPOINT, 101
$
$ USE TRANSFER FUNCTIONS TO TRACK DIFFERENCES
$ E101= UG1DOF2 - UG3DOF2
$
TF, 100, 101, 0, 1., 0., 0.,
, 1, 2, -1., 0., 0.,
, 3, 2, 1., 0., 0.
$
```



- This could also have been done using MPCs, but TF were used for this example since that is the topic of this section.

Nonlinear Transient Force

- **Grid point-oriented nonlinearity**

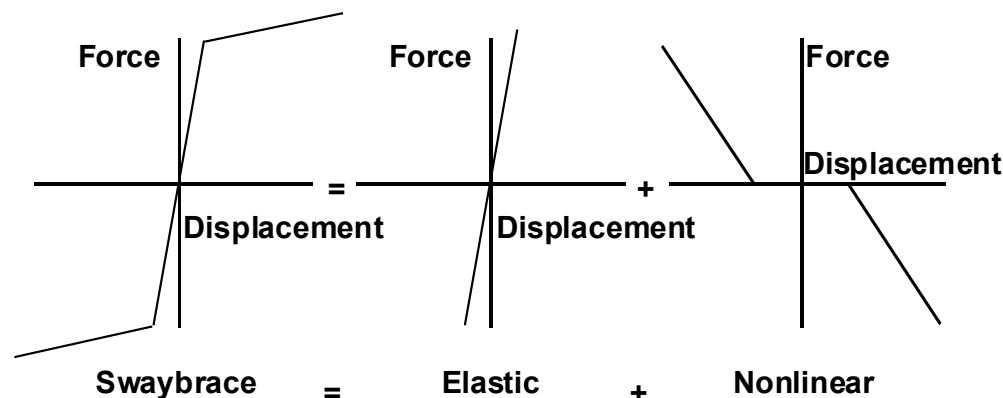
$$\mathbf{M}\ddot{\mathbf{u}}(\mathbf{t}) + \mathbf{B}\dot{\mathbf{u}}(\mathbf{t}) + \mathbf{K}\mathbf{u}(\mathbf{t}) = \mathbf{P}(\mathbf{t}) + \mathbf{N}(\mathbf{t})$$

- **N(t) is the nonlinear force and is a function of the displacement and the velocity.**
- **Defined on NOLINi Bulk Data entry that is selected by the NONLINEAR Case Control command.**
- **Available only in transient response solution sequences (must be in E-set for modal and D-set for direct formulation).**

Nonlinear Transient Force (cont.)

- Nonlinear force represents a deviation from linearity.

$$\mathbf{M}\ddot{\mathbf{u}}(\mathbf{t}) + \mathbf{B}\dot{\mathbf{u}}(\mathbf{t}) + \mathbf{K}\mathbf{u}(\mathbf{t}) = \mathbf{P}(\mathbf{t}) + \mathbf{N}(\mathbf{t})$$



- Easiest to implement in direct solutions. Apply nonlinear force directly to grid point. Modal solutions require transfer functions and extra points since only E-set points may have nonlinear loads in a modal formulation.

Considerations When Using Nonlinear Forces

- **Nonlinear forces are based on the solution at the previous time step.**

$$\mathbf{M}\ddot{\mathbf{u}}(t) + \mathbf{B}\dot{\mathbf{u}}(t) + \mathbf{K}\mathbf{u}(t) = \mathbf{P}(t) + \mathbf{N}(t - \Delta t)$$

- **A smaller Δt will give a more accurate solution.**
- **Easier to use in direct solution sequences than in modal solution sequences**
- **Nonlinear forces may be applied only to members of D-set (A-set + E-set) for direct and H-set (modal set + E-set). DOFs cannot be OMITted nor can they be dependent upon the motion of other omitted DOFs.**
- **May need a local coordinate system for the nonlinear force applied in a non-basic axis direction. Verify this with a pilot model.**

Types of NOLINs

- **NOLIN1 - Nonlinear transient load as a tabular function**

- Function of displacement $P_i(t_n) = ST(u_j(t_n))$

- Function of velocity $P_i(t) = ST(\dot{u}_j(t_n))$

where
$$\dot{u}(t_n) = \frac{u(t_n) - u(t_{n-1})}{\Delta t}$$

- **NOLIN2 - Nonlinear transient load as the product of two variables**

$$P_i(t_n) = S X_j(t_n) X_k(t_n)$$

where X_j and X_k can be either two displacements or two velocities or one of each

Types of NOLINs (cont.)

- **NOLIN3 - Nonlinear transient load as a positive variable raised to a power**

$$P_i(t_n) = \begin{cases} S(X_j(t_n))^A, & X_j(t_n) > 0 \\ 0, & X_j(t_n) \leq 0 \end{cases}$$

- where X_j can be a displacement or a velocity
- **NOLIN4 - Nonlinear transient load as a negative variable raised to a power**

$$P_i(t_n) = \begin{cases} -S(-X_j(t_n))^A, & X_j(t_n) < 0 \\ 0, & X_j(t_n) \geq 0 \end{cases}$$

- where X_j can be a displacement or a velocity

WORKSHOP 3

LINEAR TRANSIENT ANALYSIS USING NOLINS

SECTION 8

Test-Analysis Correlation

Introduction to Test-Analysis Correlation

- **MSC Nastran results and test data may not match due to modeling and testing uncertainties.**
- **Common saying: “No one believes the analytical results (except for the modeler), and everyone believes the test data (except for the experimentalist).”**
- **Sources of modeling uncertainties:**
 - Physics of the problem being simulated
 - Boundary conditions
 - Material properties
 - Joint flexibility
 - “As-built” versus “as-designed”
 - Damping

Introduction to Test-Analysis Correlation (cont.)

- **Goals of test-analysis correlation:**
 - Assess the degree of correlation between MSC Nastran results and test data.
 - Refine the MSC Nastran model to match test data.
- **Person doing the correlation must understand the test data, the MSC Nastran results, and the uncertainties in both.**

Integrated Test-Analysis Procedure

- **Four phases of an integrated test-analysis procedure:**
 1. Pre-test planning (test simulation)
 2. Data acquisition. (Capture raw data, such as, accelerations.)
 3. Data reduction and analysis. (Process raw data into quantities of interest, such as, mode shapes.)
 4. Post-test evaluation. (Assess goodness of fit between test data and MSC Nastran results; refine the MSC Nastran model to match the test data.)
- **The analyst is involved in phases 1 and 4, and the experimentalist is involved in phases 2 and 3.**

Pre-Test Planning

- **Create a baseline MSC Nastran model to determine optimal excitation and measurement locations. There are two methods for doing this:**
 - Simulation and inspection
 - Cross-orthogonality check
- **Simulation and inspection.**
 - Use MSC Nastran to simulate the test, and choose input and output locations that give maximum response.
- **Cross-orthogonality check:**
 - From a proposed set of measurement locations create an A-set, use Guyan reduction, and compute the mode shapes normalizing to a unit modal mass. Call this set of vectors (Φ_t , as in “test” modes). Output the modes that span the frequency range of interest and the A-set mass matrix.
 - Then, remove the A-set, repeat the modal calculation computing responses for the A-set DOFs for the full model, and output these modes (Φ_a , as in “analysis” modes).

Selection of ASET Points

- **The following guidelines can be used as the initial selection of ASET DOFs (potential accelerometer locations)**
 - Spread the grid point throughout the structure to capture the critical modes
 - Include large mass locations
 - Use the GPKE (grid point kinetic energy) output as a guideline and pick points that have high kinetic energy
 - The above may be limited by number of available instrumentation points
 - The above locations can be adjusted based on subsequent orthogonality check (see later)

Pre-Test Planning (cont.)

- In a third MSC Nastran run, read both sets of results and compute:

$$\Phi_t^T M_a \Phi_a$$

- If the proposed measurement locations (A-set) are adequate, then the resulting matrix has 1 on the diagonal and 0 as off-diagonal terms. If the off-diagonal terms are not 0, the proposed measurement locations are not adequate and a new set must be formulated. (In actuality, off-diagonal terms less than 0.05 are acceptable.)
- DMAP alter for pre-test planning, premaca.alt is provided with each version of the MSC Nastran delivery.

Pre-Test Planning (cont.)

- **Modal effective mass⁽¹⁾ and modal kinetic energy⁽²⁾ calculations can also be made in MSC Nastran to ensure that the test specimen is well understood before testing.**
 - **Test and analytical locations need to align fully (location and coordinate direction) to facilitate pre-test planning and post-test evaluation. RBARs, MPCs, and alternate output coordinate systems can be used to “line up” the locations.**
 - **Once the excitation and measurement locations are verified, then the test should be simulated to ensure that the test specimen is not overstressed during testing.**
1. **For the i-th mode, effective mass = $(\Phi_i^T M D_m)^2$**
where M = mass matrix and Dm = rigid-body vector.
 2. **For the i-th mode, kinetic energy = $M \Phi_i^2$. This shows the energy distribution within a mode.**

Post-Test Evaluation

- **Compare MSC Nastran results to test data by inspection, cross-orthogonality or MAC checks.**
- **Inspection—graphics**
 - XY plots:
 - Plot test data and MSC Nastran results together and assess the degree of correlation in magnitude and frequency content.
 - Make sure that the curves plotted together represent the same spatial location and direction.
 - Structure plots:
 - Plot test and analytical mode shapes together and assess the degree of correlation. (Plots can also be animated.)
- **Inspection can also be applied to non-graphical data such as, comparison of measured and computed resonant frequencies.**

Post-Test Evaluation (cont.)

- **Cross-orthogonality check**

- Quantitative check on the degree of test-analysis correlation is the cross-orthogonality check defined by

$$\Phi_t^T M_{aa} \Phi_a = \text{COR}$$

where Φ_t = test modes
 M_{aa} = A-set mass matrix
 Φ_a = analysis modes computed for the A-set

- Off-diagonal terms should be less than 10% of the diagonal terms in order to have a reliable match between test and analysis.
 - Modes may be “switched” which is reflected by large off-diagonal terms in COR, the correlation matrix.
 - Note that the reduced mass matrix is influential in the results
- **If there are discrepancies between test data and analytical results, it may be desirable to refine the MSC Nastran model to get a better match.**

Post-Test Evaluation (cont.)

- **MAC (Modal Assurance Criteria) check**

- MAC is a measure of the independence of one eigenvector with respect to another.

$$MAC_{ij} = \frac{[\{\Phi_i\}^T \{\Psi_j\}]^2}{[\{\Phi_i\}^T \{\Phi_i\}][\{\Psi_j\}^T \{\Psi_j\}]}$$

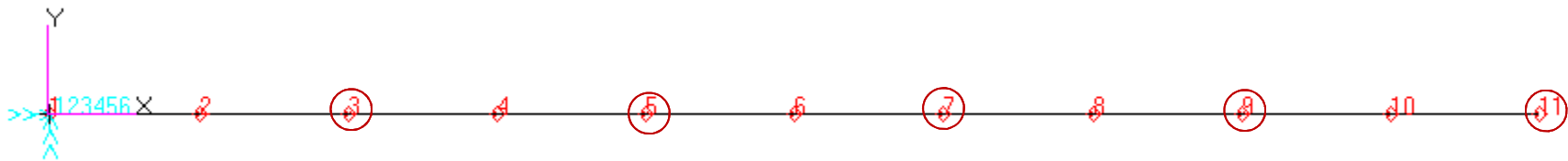
where Φ_i = test modes

Ψ_j = analysis modes computed for the A-set

- Off-diagonal terms should be less than 10% of the diagonal terms in order to have a reliable match between test and analysis.
 - Modes may be “switched” which is reflected by large off-diagonal terms in COR, the correlation matrix.
 - Note there is no mass in the equation... the results are based only on the difference in the eigenvectors
- **If there are discrepancies between test data and analytical results, it may be desirable to refine the MSC Nastran model to get a better match.**

Case Study – Problem Description

- Consider the 2-D beam model shown below (all grid's 345 DOF are constrained). Assume that accelerometers are placed on the beam at every other grid point. The translational accelerations are measured in the x and y directions.
- It is desired to match test data at the accelerometer locations using this Finite Element model.



○ Represents grids placed in the ASET

Case Study – Pre Test Processing

- **The first step is to evaluate the selection of the ASET (accelerometer) location's capability of representing the full structure's mode shapes.**
- **This can be done using the following 2 runs**
 - Run a normal modes run of the full FEM using the SSSALTER file pchdispa.alt to punch out the full model eigenvectors
 - Run a second normal modes run identifying the desired ASET DOF and use the SSSALTER file premaca.alt to calculate the MAC and Cross-Orthogonality of the ASET modes relative to the full model modes

What is DMAP?

- **The SSSALTER location shown on the previous slide provide MSC-developed DMAPs**
- **DMAP allows the user to modify the standard Nastran solution sequence to do special operations**
- **Direct Matrix Abstraction Programming (DMAP)**
 - Similar to FORTRAN
 - Provides full access to the functions of MSC Nastran
 - The language used for all solutions in MSC Nastran
- **DMAP allows users to develop their own custom routines within the Nastran code**
- **See the DMAP Programmer's Guide for more details, or the NAS110 Course**

What Can I Do With DMAP?

- **Modify the MSC-provided solutions**
 - add additional output
 - calculate additional quantities
 - change the way the problem is solved
- **Access the MSC database**
 - Store information for later use
 - Retrieve information for use in other applications
- **Run external programs (spawn a process) during the MSC.Nastran execution**

Case Study – Pre Test Processing, Step 1

- The input file, step1_punch_full_model.dat is shown below:

```
$ TEST-ANALYSIS, Step 1
$ punch out full model eigenvectors
SOL 103
diag 8,15
COMPILE MODERS, SOUIN=MSCSOU, NOLIST, NOREF
$
$ INCLUDE ALTER TO PUNCH DISPLACEMENTS
$
INCLUDE 'SSSALTERDIR:pchdispa.alt'
$
CEND
TITLE = TEST-ANALYSIS: punch analysis eigenvectors
SUBTITLE = FULL MODEL
LABEL = ALL POINTS
DISP = ALL
SPC=1
METHOD=100
BEGIN BULK
PARAM POST 0
PARAM AUTOSPC YES
GRDSET, , , , , , 345
BAROR, , , , , 0., 1., 0.
```

```
GRID 1 0.0 0.0 0.0
GRID 2 10. 0.0 0.0
GRID 3 20. 0.0 0.0
GRID 4 30. 0.0 0.0
GRID 5 40. 0.0 0.0
GRID 6 50. 0.0 0.0
GRID 7 60. 0.0 0.0
GRID 8 70. 0.0 0.0
GRID 9 80. 0.0 0.0
GRID 10 90. 0.0 0.0
GRID 11 100. 0.0 0.0
CBAR 1 1 1 2
CBAR 2 1 2 3
CBAR 3 1 3 4
CBAR 4 1 4 5
CBAR 5 1 5 6
CBAR 6 1 6 7
CBAR 7 1 7 8
CBAR 8 1 8 9
CBAR 9 1 9 10
CBAR 10 1 10 11
SPC 1 1 123456 0.0
PBAR 1 1 1.0 160.0 160.0
MAT1 1 3.+7 .3 7.7E-4
EIGRL 100 0.0 10000. 6
ENDDATA
```


Case Study – Pre Test Processing, Step 1 (cont.)

- The resulting eigenvalues are shown below... these are considered the ‘best’ FEM answer at this time and will be compared against the ASET selection based answers in Step 2.
 - Note that at this point, only the full FEM is being compared against the reduced FEM to evaluate the ASET selection.

MODE NO.	EXTRACTION ORDER	EIGENVALUE	R E A L E I G E N V A L U E S		GENERALIZED MASS	GENERALIZED STIFFNESS
			RADIANS	CYCLES		
1	1	7.636173E+05	8.738520E+02	1.390779E+02	1.000000E+00	7.636173E+05
2	2	9.593501E+06	3.097338E+03	4.929566E+02	1.000000E+00	9.593501E+06
3	3	2.932654E+07	5.415398E+03	8.618873E+02	1.000000E+00	2.932654E+07
4	4	8.492998E+07	9.215746E+03	1.466731E+03	1.000000E+00	8.492998E+07
5	5	2.253432E+08	1.501144E+04	2.389144E+03	1.000000E+00	2.253432E+08
6	6	2.282285E+08	1.510723E+04	2.404391E+03	1.000000E+00	2.282285E+08

Case Study – Pre Test Processing, Step 2

- The input file, **step2_premac.dat** is shown below
 - Note that this is the same FEM as Step 1, with the addition of ASET entries to identify the grids representing the test accelerometer locations, an include for the full model punched eigenvectors, and an include for the correlation alter.

```
$ Test-Analysis, Step 2
$ compare the ASET selected DOF against
$ the full FEM eigenvectors
SOL 103
diag 8,15
$
$ INCLUDE ALTER FOR CROSS-ORTHOGONALITY CHECKS
$
INCLUDE 'SSSALTERDIR:premaca.alt'
$
CEND
ECHO=NONE
TITLE = TEST-ANALYSIS: ANALYSIS ASET Checkout
SUBTITLE = REDUCED MODEL
LABEL = 5 POINTS
DISP = ALL
SPC=1
METHOD=100
BEGIN BULK
$
$ READ PCH FILE OF FULL MODEL
$ INCLUDE PUNCHED DISPLS FROM Step 1, FULL RUN
INCLUDE 'step1_punch_full_model.pch'
$
PARAM POST 0
PARAM AUTOSPC YES
GRDSET, , , , , , 345
BAROR, , , , , 0., 1., 0.
```

```
GRID 1 0.0 0.0 0.0
GRID 2 10. 0.0 0.0
GRID 3 20. 0.0 0.0
GRID 4 30. 0.0 0.0
GRID 5 40. 0.0 0.0
GRID 6 50. 0.0 0.0
GRID 7 60. 0.0 0.0
GRID 8 70. 0.0 0.0
GRID 9 80. 0.0 0.0
GRID 10 90. 0.0 0.0
GRID 11 100. 0.0 0.0
CBAR 1 1 1 2
CBAR 2 1 2 3
CBAR 3 1 3 4
CBAR 4 1 4 5
CBAR 5 1 5 6
CBAR 6 1 6 7
CBAR 7 1 7 8
CBAR 8 1 8 9
CBAR 9 1 9 10
CBAR 10 1 10 11
$ REDUCE TO TEST DOF
ASET1, 12, 3, 5, 7, 9, 11
SPC 1 1 123456 0.0
PBAR 1 1 1.0 160.0 160.0
MAT1 1 3.+7 .3 7.7E-4
EIGRL 100 0.0 10000. 6
ENDDATA
```

Case Study – Pre Test Processing, Step 2 (cont.)

- The resulting eigenvalues are shown below... these are compared against the 'full' FEM results
- ASET based modes:

MODE NO.	EXTRACTION ORDER	EIGENVALUE	R E A L E I G E N V A L U E S		GENERALIZED MASS	GENERALIZED STIFFNESS
			RADIANS	CYCLES		
1	1	7.636363E+05	8.738629E+02	1.390796E+02	1.000000E+00	7.636363E+05
2	2	9.652552E+06	3.106856E+03	4.944714E+02	1.000000E+00	9.652552E+06
3	3	2.935258E+07	5.417802E+03	8.622700E+02	1.000000E+00	2.935258E+07
4	4	8.952774E+07	9.461910E+03	1.505910E+03	1.000000E+00	8.952774E+07
5	5	2.267729E+08	1.505898E+04	2.396711E+03	1.000000E+00	2.267729E+08
6	6	2.597403E+08	1.611646E+04	2.565014E+03	1.000000E+00	2.597403E+08

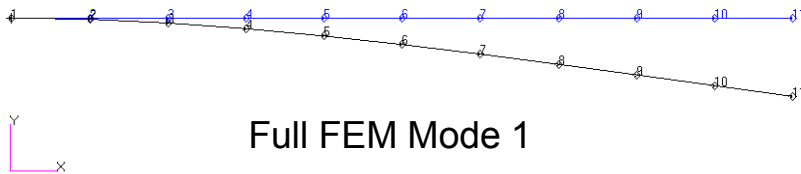
- Full FEM modes:

MODE NO.	EXTRACTION ORDER	EIGENVALUE	R E A L E I G E N V A L U E S		GENERALIZED MASS	GENERALIZED STIFFNESS
			RADIANS	CYCLES		
1	1	7.636173E+05	8.738520E+02	1.390779E+02	1.000000E+00	7.636173E+05
2	2	9.593501E+06	3.097338E+03	4.929566E+02	1.000000E+00	9.593501E+06
3	3	2.932654E+07	5.415398E+03	8.618873E+02	1.000000E+00	2.932654E+07
4	4	8.492998E+07	9.215746E+03	1.466731E+03	1.000000E+00	8.492998E+07
5	5	2.253432E+08	1.501144E+04	2.389144E+03	1.000000E+00	2.253432E+08
6	6	2.282285E+08	1.510723E+04	2.404391E+03	1.000000E+00	2.282285E+08

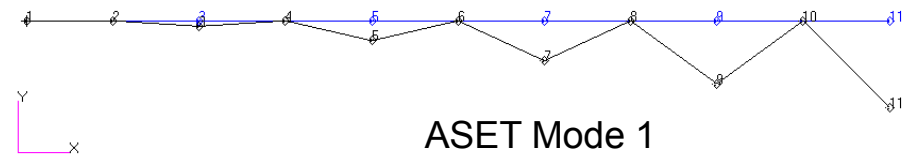
- As can be seen, the frequencies are pretty close... but it is important to also compare the mode shapes to ensure these modes represent the same mode of vibration

Case Study – Pre Test Processing, Step 2 (cont.)

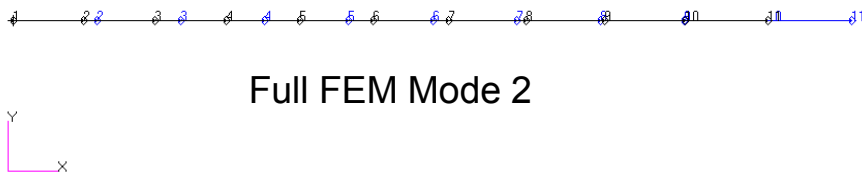
- **It is good to perform a visual comparison of the mode shapes**
 - The ASET modes only have movement of grids 3, 5, 7, 9, 11- so it makes it more difficult to visualize the deformation... just visually 'interpolate' the missing motion



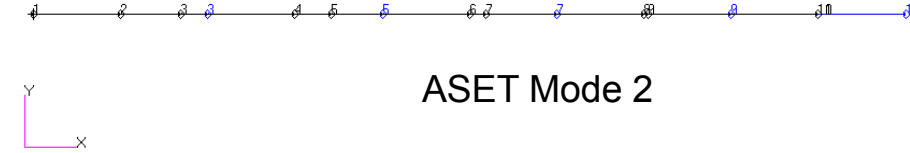
Full FEM Mode 1



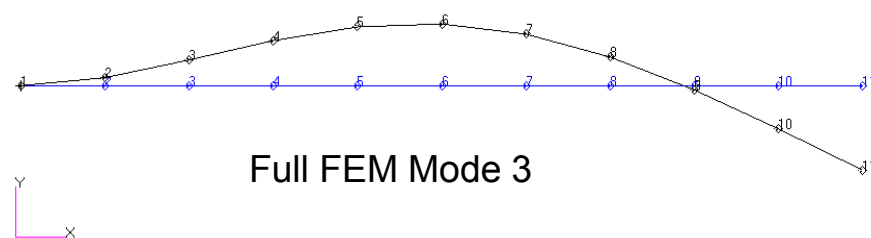
ASET Mode 1



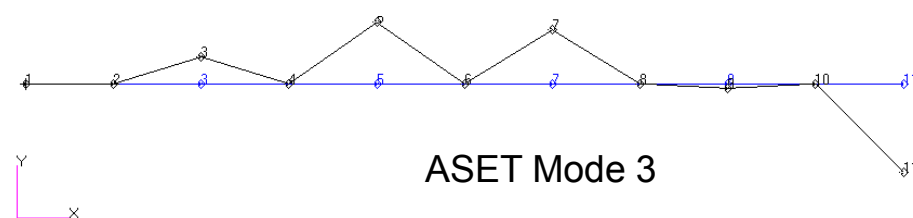
Full FEM Mode 2



ASET Mode 2



Full FEM Mode 3



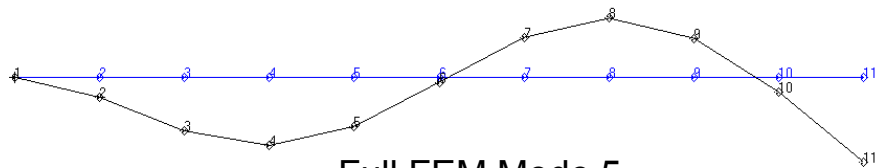
ASET Mode 3

Case Study – Pre Test Processing, Step 2 (cont.)

- **It is also good to perform a visual comparison of the mode shapes (cont.)**
 - As seen here, and for modes 1-3 on the previous slide, these modes represent the same type of deformation... but a better comparison uses MAC and XORTHO checks



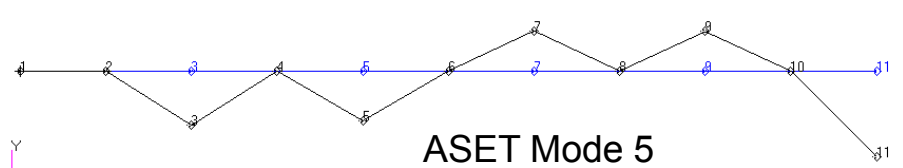
Full FEM Mode 4



Full FEM Mode 5



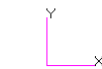
ASET Mode 4



ASET Mode 5



Full FEM Mode 6



ASET Mode 6

Case Study – Pre Test Processing, Step 2 (cont.)

MSC Nastran Frequencies and Mode Shapes

Grid Point/ DOF	Mode 1	Mode 2	Mode 3	Mode 4	Mode 5	Mode 6
3x	0.00	-1.58	0.00	-4.35	0.00	-5.88
3y	0.46	0.00	2.13	0.00	-4.30	0.00
5x	0.00	-3.01	0.00	-5.12	0.00	0.00
5y	1.65	0.00	4.87	0.00	-3.93	0.00
7x	0.00	-4.15	0.00	-1.66	0.00	5.88
7y	3.31	0.00	4.27	0.00	3.18	0.00
9x	0.00	-4.88	0.00	3.16	0.00	0.00
9y	5.20	0.00	-0.36	0.00	3.11	0.00
11x	0.00	-5.13	0.00	5.38	0.00	-5.88
11y	7.18	0.00	-7.02	0.00	-6.82	0.00
Frequency (Hz)	139.1	494.5	862.5	1505.9	2403.5	2565

Case Study – Pre Test Processing, Step 2 (cont.)

- **Additional output from Step 2**
 - MAC results (note the diagonal terms are 1.0 and off diagonal $\sim < .1$)

```

^^^MODAL ASSURANCE CRITERIA FOR RESIDUAL STRUCTURE A-SET
^^^
^^^MODAL ASSURANCE CALCULATION IS PERFORMED FOR      6      CALCULATED MODES AND      6      INPUT MODES
^^^
^^^ANALYTICAL MODE      1      COMPARED TO INPUT MODE 1      - MAC =      1.000000E+00
^^^ANALYTICAL MODE      1      COMPARED TO INPUT MODE 2      - MAC =      4.993866E-34
^^^ANALYTICAL MODE      1      COMPARED TO INPUT MODE 3      - MAC =      9.598562E-02
^^^ANALYTICAL MODE      1      COMPARED TO INPUT MODE 4      - MAC =      1.548841E-30
^^^ANALYTICAL MODE      1      COMPARED TO INPUT MODE 5      - MAC =      1.005941E-01
^^^ANALYTICAL MODE      1      COMPARED TO INPUT MODE 6      - MAC =      1.074457E-26
^^^ANALYTICAL MODE      2      COMPARED TO INPUT MODE 1      - MAC =      4.007434E-33
^^^ANALYTICAL MODE      2      COMPARED TO INPUT MODE 2      - MAC =      9.999999E-01
^^^ANALYTICAL MODE      2      COMPARED TO INPUT MODE 3      - MAC =      6.834812E-33
^^^ANALYTICAL MODE      2      COMPARED TO INPUT MODE 4      - MAC =      2.777777E-02
^^^ANALYTICAL MODE      2      COMPARED TO INPUT MODE 5      - MAC =      1.654639E-27
^^^ANALYTICAL MODE      2      COMPARED TO INPUT MODE 6      - MAC =      2.777777E-02
^^^ANALYTICAL MODE      3      COMPARED TO INPUT MODE 1      - MAC =      9.622484E-02
^^^ANALYTICAL MODE      3      COMPARED TO INPUT MODE 2      - MAC =      7.890231E-34
^^^ANALYTICAL MODE      3      COMPARED TO INPUT MODE 3      - MAC =      9.999998E-01
^^^ANALYTICAL MODE      3      COMPARED TO INPUT MODE 4      - MAC =      6.198319E-30
^^^ANALYTICAL MODE      3      COMPARED TO INPUT MODE 5      - MAC =      1.059392E-01
^^^ANALYTICAL MODE      3      COMPARED TO INPUT MODE 6      - MAC =      1.186206E-26
^^^ANALYTICAL MODE      4      COMPARED TO INPUT MODE 1      - MAC =      1.113581E-32
^^^ANALYTICAL MODE      4      COMPARED TO INPUT MODE 2      - MAC =      2.777777E-02
^^^ANALYTICAL MODE      4      COMPARED TO INPUT MODE 3      - MAC =      2.982440E-34
^^^ANALYTICAL MODE      4      COMPARED TO INPUT MODE 4      - MAC =      1.000000E+00
^^^ANALYTICAL MODE      4      COMPARED TO INPUT MODE 5      - MAC =      1.547674E-27
^^^ANALYTICAL MODE      4      COMPARED TO INPUT MODE 6      - MAC =      2.777777E-02
^^^ANALYTICAL MODE      5      COMPARED TO INPUT MODE 1      - MAC =      1.016641E-01
^^^ANALYTICAL MODE      5      COMPARED TO INPUT MODE 2      - MAC =      5.602917E-32
^^^ANALYTICAL MODE      5      COMPARED TO INPUT MODE 3      - MAC =      1.071091E-01
^^^ANALYTICAL MODE      5      COMPARED TO INPUT MODE 4      - MAC =      4.712732E-31
^^^ANALYTICAL MODE      5      COMPARED TO INPUT MODE 5      - MAC =      9.999937E-01
^^^ANALYTICAL MODE      5      COMPARED TO INPUT MODE 6      - MAC =      1.041557E-25
^^^ANALYTICAL MODE      6      COMPARED TO INPUT MODE 1      - MAC =      1.734602E-31
^^^ANALYTICAL MODE      6      COMPARED TO INPUT MODE 2      - MAC =      2.777778E-02
^^^ANALYTICAL MODE      6      COMPARED TO INPUT MODE 3      - MAC =      1.814836E-31
^^^ANALYTICAL MODE      6      COMPARED TO INPUT MODE 4      - MAC =      2.777778E-02
^^^ANALYTICAL MODE      6      COMPARED TO INPUT MODE 5      - MAC =      6.200561E-26
^^^ANALYTICAL MODE      6      COMPARED TO INPUT MODE 6      - MAC =      1.000000E+00

```

Case Study – Pre Test Processing, Step 2 (cont.)

- Additional output from Step 2 (cont.)

- CHECKIT

- This represents $(\Phi_{FEM})^T * M_{ASET} * \Phi_{FEM}$ and is to ensure the reduced mass isn't causing any inaccuracies on its own
- It should have 1.0 on the diagonal and very near zero on the off diagonal

```

0      MATRIX CHECKIT (GINO NAME 101 ) IS A DB PREC      6 COLUMN X      6 ROW SYMMETRIC MATRIX.
0COLUMN      1      ROWS      1 THRU      6      -----
ROW
1) 1.0000D+00 -5.9941D-17 -4.3685D-04 -2.1899D-15 -1.3560D-03 -4.1608D-16
0COLUMN      2      ROWS      1 THRU      6      -----
ROW
1) -5.9941D-17 1.0000D+00 -2.4441D-17 5.4149D-11 -6.9224D-16 -3.4198D-11
0COLUMN      3      ROWS      1 THRU      6      -----
ROW
1) -4.3685D-04 -2.4441D-17 1.0000D+00 2.9105D-15 -1.5855D-03 3.3187D-15
0COLUMN      4      ROWS      1 THRU      6      -----
ROW
1) -2.1899D-15 5.4149D-11 2.9105D-15 1.0000D+00 -2.3607D-15 -3.5886D-11
0COLUMN      5      ROWS      1 THRU      6      -----
ROW
1) -1.3560D-03 -6.9224D-16 -1.5855D-03 -2.3607D-15 1.0000D+00 8.3060D-14
0COLUMN      6      ROWS      1 THRU      6      -----
ROW
1) -4.1608D-16 -3.4198D-11 3.3187D-15 -3.5886D-11 8.3060D-14 1.0000D+00
    
```


Case Study – Pre Test Processing, Step 2 (cont.)

- Additional output from Step 2 (cont.)

- ORTHOA

- This represents $(\Phi_{ASET})^T * M_{ASET} * \Phi_{FEM}$ and is to compare both the effect of the reduced mass, along with the effect of the reduced mode shapes
- It should have 1.0 on the diagonal and near zero on the off diagonal

```

0      MATRIX ORTHOA   (GINO NAME 101 ) IS A DB PREC      6 COLUMN X      6 ROW SQUARE  MATRIX.
OCOLUMN      1      ROWS      1 THRU      6      -----
ROW
1)  -1.0000D+00  -8.2993D-17  2.9384D-07  -3.8073D-17  1.3846D-08  1.1697D-17
OCOLUMN      2      ROWS      1 THRU      6      -----
ROW
1)  -2.3052D-17  1.0000D+00  5.1892D-17  -4.9003D-12  -7.5604D-17  3.4195D-11
OCOLUMN      3      ROWS      1 THRU      6      -----
ROW
1)  4.3714D-04  -7.6321D-17  1.0000D+00  8.2268D-17  2.4651D-05  4.7817D-17
OCOLUMN      4      ROWS      1 THRU      6      -----
ROW
1)  2.2280D-15  4.9249D-11  2.9918D-15  -1.0000D+00  3.7943D-17  3.5884D-11
OCOLUMN      5      ROWS      1 THRU      6      -----
ROW
1)  1.3560D-03  -6.1651D-16  -1.6108D-03  2.3968D-15  1.0000D+00  2.4484D-13
OCOLUMN      6      ROWS      1 THRU      6      -----
ROW
1)  4.0439D-16  -2.6645D-15  3.3583D-15  1.8041D-15  3.2790D-13  -1.0000D+00
    
```

Case Study – Test Modes

- Since the selected ASET DOF showed excellent correlation with the full FEM, those ASET locations are used as the test accelerometer locations.
- The test data acquisition resulted in the following test frequencies and mode shapes.
- For use in subsequent test-analysis correlation, these test mode shapes need to be put into DMIG format, and named PHITEST.

Test Frequencies and Mode Shapes

Grid Point/ DOF	Mode 1	Mode 2	Mode 3	Mode 4	Mode 5	Mode 6
3x	0	0.3	0	-0.8	1	0
3y	0.05	0	-0.3	0	0	0.65
5x	0	0.6	0	-0.95	0	0
5y	0.25	0	-0.7	0	0	0.6
7x	0	0.8	0	-0.3	-1	0
7y	0.45	0	-0.6	0	0	-0.45
9x	0	0.95	0	0.6	0	0
9y	0.7	0	0.5	0	0	-0.45
11x	0	1	0	1	1	0
11y	1	0	1	0	0	1
Frequency (Hz)	137.6	501.1	875.7	1600.2	2552.6	2611.9

Case Study - Post Test Evaluation, Step 3

- Now we have a reduced FEM represented by the ASET modes
- We have Test data (mode shapes)
- The alter postmaca.alt is used with a Stand-Alone DMAP to calculate the MAC/XORTHO results for the Full FEM compared to Test and ASET FEM compared to Test

Case Study - Post Test Evaluation, Step 3 (cont.)

- Here is the input file for the FEM comparison with the Test results
- The necessary ALTER is referenced from the MSC NASTRAN installation
- The punch file from the Step 2 run is included
 - This file has the Full FEM mode shapes, the ASET FEM mode shapes and the ASET FEM Mass
- The user-created punch file containing the Test mode shapes is included
- GRID entries are added for the ASET grids

```
$ TEST-ANALYSIS, POST TEST Correlation
SOL 100
diag 8, 15
$
$ INCLUDE ALTER FOR CROSS-ORTHOGONALITY CHECKS
$
INCLUDE 'SSSALTERDIR:postmaca.alt'
$
CEND
TITLE = TEST-ANALYSIS: POST POSTMAC1
SUBTITLE = CORRELATE TEST AND ANALYSIS MODES
LABEL = 5 POINTS
BEGIN BULK
$
$ READ PCH FILE OF ANALYSIS MODEL
$ FROM PRE-TEST RUN (ASET)
$
INCLUDE 'step2_premac.pch'
$
$
$ READ PCH FILE OF TEST DATA
$ FROM EXTERNAL FILE
$
INCLUDE 'test_eigenvectors.pch'
$
$ DEFINE GRIDS FOR TEST-ANALYSIS
$ The position doesn't matter, just the ID's
$
GRID 3
GRID 5
GRID 7
GRID 9
GRID 11
$
ENDDATA
```

Case Study - Post Test Evaluation, Step 3 (cont.)

- Here is the punch file for the Test eigenvectors

```

DMIG    PHITEST    0      9      2      0
DMIG*   PHITEST    1      0
*        3      1 0.0D+00
*        3      2 0.05D+00
*        5      1 0.D+00
*        5      2 0.25D+00
*        7      1 0.D+00
*        7      2 0.45D+00
*        9      1 0.D+00
*        9      2 0.70D+00
*       11      1 0.D+00
*       11      2 1.0D+00
DMIG*   PHITEST    2      0
*        3      1 0.30D+00
*        3      2 0.D+00
*        5      1 0.60D+00
*        5      2 0.D+00
*        7      1 0.80D+00
*        7      2 0.D+00
*        9      1 0.95D+00
*        9      2 0.D+00
*       11      1 1.0D+00
*       11      2 0.D+00
DMIG*   PHITEST    3      0
*        3      1 0.D+00
*        3      2-0.30D+00
*        5      1 0.D+00
*        5      2-0.70D+00
*        7      1 0.D+00
*        7      2-0.60D+00
*        9      1 0.D+00
*        9      2 0.50D+00
*       11      1 0.D+00
*       11      2 1.0D+00

```

```

DMIG*   PHITEST    4      0
*        3      1-0.80D+00
*        3      2 0.D+00
*        5      1-0.95D+00
*        5      2 0.D+00
*        7      1-0.30D+00
*        7      2 0.D+00
*        9      1 0.60D+00
*        9      2 0.D+00
*       11      1 1.0D+00
*       11      2 0.D+00
DMIG*   PHITEST    5      0
*        3      1 1.0D+00
*        3      2 0.D+00
*        5      1 0.0D+00
*        5      2 0.D+00
*        7      1-1.0D+00
*        7      2 0.D+00
*        9      1 0.0D+00
*        9      2 0.D+00
*       11      1 1.0D+00
*       11      2 0.D+00
DMIG*   PHITEST    6      0
*        3      1 0.D+00
*        3      2 0.65D+00
*        5      1 0.D+00
*        5      2 0.60D+00
*        7      1 0.D+00
*        7      2-0.45D+00
*        9      1 0.D+00
*        9      2-0.45D+00
*       11      1 0.D+00
*       11      2 1.0D+00

```

Case Study - Post Test Evaluation, Step 3 (cont.)

- **Output from Step 3**
 - MAC results of the **Full FEM Model** and the Test modes

Note that mode 5 and mode 6 are swapped between the test modes and the analysis modes... this is identified by the 'near 1.0' off diagonal term.

MODAL ASSURANCE CALCULATION IS PERFORMED FOR				6	FULL MODEL MODES AND		6	TEST MODES
^^^	FULL MODEL MODE	1	COMPARED TO TEST MODE	1	-	MAC =	9.993510E-01	
^^^	FULL MODEL MODE	1	COMPARED TO TEST MODE	2	-	MAC =	2.215651E-33	
^^^	FULL MODEL MODE	1	COMPARED TO TEST MODE	3	-	MAC =	2.091196E-01	
^^^	FULL MODEL MODE	1	COMPARED TO TEST MODE	4	-	MAC =	1.056176E-33	
^^^	FULL MODEL MODE	1	COMPARED TO TEST MODE	5	-	MAC =	1.284397E-34	
^^^	FULL MODEL MODE	1	COMPARED TO TEST MODE	6	-	MAC =	1.062026E-01	
^^^	FULL MODEL MODE	2	COMPARED TO TEST MODE	1	-	MAC =	2.755091E-36	
^^^	FULL MODEL MODE	2	COMPARED TO TEST MODE	2	-	MAC =	9.998970E-01	
^^^	FULL MODEL MODE	2	COMPARED TO TEST MODE	3	-	MAC =	1.010103E-36	
^^^	FULL MODEL MODE	2	COMPARED TO TEST MODE	4	-	MAC =	3.038901E-02	
^^^	FULL MODEL MODE	2	COMPARED TO TEST MODE	5	-	MAC =	2.777778E-02	
^^^	FULL MODEL MODE	2	COMPARED TO TEST MODE	6	-	MAC =	3.480348E-37	
^^^	FULL MODEL MODE	3	COMPARED TO TEST MODE	1	-	MAC =	9.629782E-02	
^^^	FULL MODEL MODE	3	COMPARED TO TEST MODE	2	-	MAC =	1.929121E-33	
^^^	FULL MODEL MODE	3	COMPARED TO TEST MODE	3	-	MAC =	9.081355E-01	
^^^	FULL MODEL MODE	3	COMPARED TO TEST MODE	4	-	MAC =	2.298510E-32	
^^^	FULL MODEL MODE	3	COMPARED TO TEST MODE	5	-	MAC =	3.656624E-36	
^^^	FULL MODEL MODE	3	COMPARED TO TEST MODE	6	-	MAC =	9.552094E-02	
^^^	FULL MODEL MODE	4	COMPARED TO TEST MODE	1	-	MAC =	1.552570E-30	
^^^	FULL MODEL MODE	4	COMPARED TO TEST MODE	2	-	MAC =	2.760769E-02	
^^^	FULL MODEL MODE	4	COMPARED TO TEST MODE	3	-	MAC =	4.221343E-30	
^^^	FULL MODEL MODE	4	COMPARED TO TEST MODE	4	-	MAC =	9.998970E-01	
^^^	FULL MODEL MODE	4	COMPARED TO TEST MODE	5	-	MAC =	2.777778E-02	
^^^	FULL MODEL MODE	4	COMPARED TO TEST MODE	6	-	MAC =	1.732344E-31	
^^^	FULL MODEL MODE	5	COMPARED TO TEST MODE	1	-	MAC =	1.092327E-01	
^^^	FULL MODEL MODE	5	COMPARED TO TEST MODE	2	-	MAC =	1.656170E-27	
^^^	FULL MODEL MODE	5	COMPARED TO TEST MODE	3	-	MAC =	4.396997E-02	
^^^	FULL MODEL MODE	5	COMPARED TO TEST MODE	4	-	MAC =	1.538004E-27	
^^^	FULL MODEL MODE	5	COMPARED TO TEST MODE	5	-	MAC =	6.274038E-26	
^^^	FULL MODEL MODE	5	COMPARED TO TEST MODE	6	-	MAC =	9.995029E-01	
^^^	FULL MODEL MODE	6	COMPARED TO TEST MODE	1	-	MAC =	1.165788E-26	
^^^	FULL MODEL MODE	6	COMPARED TO TEST MODE	2	-	MAC =	2.784739E-02	
^^^	FULL MODEL MODE	6	COMPARED TO TEST MODE	3	-	MAC =	5.084902E-27	
^^^	FULL MODEL MODE	6	COMPARED TO TEST MODE	4	-	MAC =	2.784739E-02	
^^^	FULL MODEL MODE	6	COMPARED TO TEST MODE	5	-	MAC =	1.000000E+00	
^^^	FULL MODEL MODE	6	COMPARED TO TEST MODE	6	-	MAC =	1.051063E-25	

Case Study - Post Test Evaluation, Step 3 (cont.)

- Additional output from Step 3**

- MAC results of the **ASET FEM Model** and the Test modes

- Again, we see that mode 5 and mode 6 are swapped between the test modes and the analysis modes

MODAL ASSURANCE CALCULATION IS PERFORMED FOR 6 ANALYSIS MODES AND 6 REDUCED MODEL MODES			
ANALYTICAL MODE	1	COMPARED TO TEST MODE 1	- MAC = 9.993510E-01
ANALYTICAL MODE	1	COMPARED TO TEST MODE 2	- MAC = 4.279361E-34
ANALYTICAL MODE	1	COMPARED TO TEST MODE 3	- MAC = 2.091194E-01
ANALYTICAL MODE	1	COMPARED TO TEST MODE 4	- MAC = 3.849048E-35
ANALYTICAL MODE	1	COMPARED TO TEST MODE 5	- MAC = 2.365835E-35
ANALYTICAL MODE	1	COMPARED TO TEST MODE 6	- MAC = 1.062026E-01
ANALYTICAL MODE	2	COMPARED TO TEST MODE 1	- MAC = 2.333624E-34
ANALYTICAL MODE	2	COMPARED TO TEST MODE 2	- MAC = 9.998971E-01
ANALYTICAL MODE	2	COMPARED TO TEST MODE 3	- MAC = 2.315568E-33
ANALYTICAL MODE	2	COMPARED TO TEST MODE 4	- MAC = 3.038901E-02
ANALYTICAL MODE	2	COMPARED TO TEST MODE 5	- MAC = 2.777778E-02
ANALYTICAL MODE	2	COMPARED TO TEST MODE 6	- MAC = 3.315157E-33
ANALYTICAL MODE	3	COMPARED TO TEST MODE 1	- MAC = 9.653700E-02
ANALYTICAL MODE	3	COMPARED TO TEST MODE 2	- MAC = 7.442814E-34
ANALYTICAL MODE	3	COMPARED TO TEST MODE 3	- MAC = 9.082690E-01
ANALYTICAL MODE	3	COMPARED TO TEST MODE 4	- MAC = 5.214750E-33
ANALYTICAL MODE	3	COMPARED TO TEST MODE 5	- MAC = 8.916561E-34
ANALYTICAL MODE	3	COMPARED TO TEST MODE 6	- MAC = 9.556985E-02
ANALYTICAL MODE	4	COMPARED TO TEST MODE 1	- MAC = 5.781594E-33
ANALYTICAL MODE	4	COMPARED TO TEST MODE 2	- MAC = 2.760769E-02
ANALYTICAL MODE	4	COMPARED TO TEST MODE 3	- MAC = 1.164472E-32
ANALYTICAL MODE	4	COMPARED TO TEST MODE 4	- MAC = 9.998971E-01
ANALYTICAL MODE	4	COMPARED TO TEST MODE 5	- MAC = 2.777778E-02
ANALYTICAL MODE	4	COMPARED TO TEST MODE 6	- MAC = 2.711909E-32
ANALYTICAL MODE	5	COMPARED TO TEST MODE 1	- MAC = 1.103399E-01
ANALYTICAL MODE	5	COMPARED TO TEST MODE 2	- MAC = 5.666964E-32
ANALYTICAL MODE	5	COMPARED TO TEST MODE 3	- MAC = 4.480625E-02
ANALYTICAL MODE	5	COMPARED TO TEST MODE 4	- MAC = 3.756292E-32
ANALYTICAL MODE	5	COMPARED TO TEST MODE 5	- MAC = 2.610863E-30
ANALYTICAL MODE	5	COMPARED TO TEST MODE 6	- MAC = 9.994664E-01
ANALYTICAL MODE	6	COMPARED TO TEST MODE 1	- MAC = 1.938919E-31
ANALYTICAL MODE	6	COMPARED TO TEST MODE 2	- MAC = 2.784740E-02
ANALYTICAL MODE	6	COMPARED TO TEST MODE 3	- MAC = 7.675702E-32
ANALYTICAL MODE	6	COMPARED TO TEST MODE 4	- MAC = 2.784740E-02
ANALYTICAL MODE	6	COMPARED TO TEST MODE 5	- MAC = 1.000000E+00
ANALYTICAL MODE	6	COMPARED TO TEST MODE 6	- MAC = 2.168827E-30

Case Study - Post Test Evaluation, Step 3 (cont.)

- Additional output from Step 3 (cont.)

- CHECKIT

- This represents $(\Phi_{\text{test}})^T * M_{\text{ASET}} * \Phi_{\text{test}}$ and is to check the quality of the test modes
- It should have 1.0 on the diagonal and less than .1 on the off diagonal
 - As can be seen, the diagonal is very good, but the off-diagonal has some higher than desired values

0	MATRIX CHECKIT	(GINO NAME 101)	IS A DB	PREC	6 COLUMN X	6 ROW SQUARE	MATRIX.
0COLUMN	1	ROWS	1 THRU	6	-----		
ROW							
1)	1.0000D+00	0.0000D+00	2.4999D-01	0.0000D+00	0.0000D+00	2.7419D-02	
0COLUMN	2	ROWS	2 THRU	5	-----		
ROW							
2)	1.0000D+00	0.0000D+00	8.5251D-03	3.0224D-11			
0COLUMN	3	ROWS	1 THRU	6	-----		
ROW							
1)	2.4999D-01	0.0000D+00	1.0000D+00	0.0000D+00	0.0000D+00	-1.2581D-01	
0COLUMN	4	ROWS	2 THRU	5	-----		
ROW							
2)	8.5251D-03	0.0000D+00	1.0000D+00	3.1716D-11			
0COLUMN	5	ROWS	2 THRU	5	-----		
ROW							
2)	3.0224D-11	0.0000D+00	3.1716D-11	1.0000D+00			
0COLUMN	6	ROWS	1 THRU	6	-----		
ROW							
1)	2.7419D-02	0.0000D+00	-1.2581D-01	0.0000D+00	0.0000D+00	1.0000D+00	

Case Study - Post Test Evaluation, Step 3 (cont.)

- Additional output from Step 3 (cont.)

- ORTHO2

- This represents $(\Phi_{\text{test}})^T * M_{\text{ASET}} * \Phi_{\text{FEM}}$ and is to compare the test mode shapes with the FULL FEM mode shapes partitioned to the ASET DOF
- It should have >.95 on the diagonal and <.1 on the off diagonal
 - The criteria for what is appropriate for the diagonal and off diagonal is set by each organization, .95/.10 is commonly used... since this is comparing the FULL FEM to the test, the results will be worse
 - As can be seen, the diagonal is pretty good, but the off-diagonal has a higher than desired value
 - The swapped modes are also evident in the XORTHO output, as seen previously in the MAC

```

^^^RESULTS OF CROSS-ORTHOGONALITY TEST
^^^EACH ROW REPRESENTS ONE FULL MODEL MODE, EACH COLUMN REPRESENTS AN INPUT MODE
0 MATRIX ORTHO2 (GINO NAME 101 ) IS A DB PREC 6 COLUMN X 6 ROW SQUARE MATRIX.
0 COLUMN 1 ROWS 1 THRU 6
1) -9.9962D-01 -1.4401D-18 6.0394D-03 2.1261D-15 -1.1755D-02 -3.4128D-15
0 COLUMN 2 ROWS 1 THRU 6
1) 6.1428D-17 -9.9383D-01 2.4584D-17 7.7971D-04 6.6532D-16 2.6177D-11
0 COLUMN 3 ROWS 1 THRU 6
1) -2.5888D-01 -6.3331D-19 -9.4030D-01 -2.1756D-15 1.0828D-01 2.8097D-14
0 COLUMN 4 ROWS 1 THRU 6
1) -3.9951D-17 -9.3253D-03 1.5905D-16 -9.4702D-01 2.5009D-15 2.7468D-11
0 COLUMN 5 ROWS 1 THRU 6
1) -1.2604D-17 -6.3978D-11 -2.6892D-17 -6.3978D-11 -2.4342D-13 8.6603D-01
0 COLUMN 6 ROWS 1 THRU 6
1) -1.4139D-02 1.1662D-22 2.3332D-02 -5.5994D-17 -9.8756D-01 -2.8514D-13
    
```

Case Study - Post Test Evaluation, Step 3 (cont.)

- **Additional output from Step 3 (cont.)**

- ORTHOA

- This represents $(\Phi_{\text{test}})^T * M_{\text{ASET}} * \Phi_{\text{ASET}}$ and is to compare the test mode shapes with the ASET FEM mode shapes
 - It should have >.95 on the diagonal and <.1 on the off diagonal
 - The criteria for what is appropriate for the diagonal and off diagonal is set by each organization, .95/.10 is commonly used... this will often be better then the FULL FEM comparison, as it more closely represents the test data acquisition
 - As can be seen, the diagonal is very good and again, the off-diagonal has one out-of-bound term
 - The swapped modes are also evident in the XORTHO output, as seen previously in the MAC

```

^^^RESULTS OF CROSS-ORTHOGONALITY TEST
^^^EACH ROW REPRESENTS ONE ANALYTICAL MODE, EACH COLUMN REPRESENTS AN INPUT MODE
0  MATRIX ORTHOA  (GINO NAME 101 ) IS A DB  PREC  6 COLUMN X  6 ROW SQUARE  MATRIX.
0COLUMN  1  ROWS  1 THRU  6  -----
ROW
1)  9.9966D-01  2.0417D-17  5.6116D-03  3.4999D-19  -1.3249D-02  1.3340D-17
0COLUMN  2  ROWS  1 THRU  6  -----
ROW
1)  2.1600D-17  -9.9996D-01  -5.1634D-17  -8.2329D-04  7.4215D-17  -3.0223D-11
0COLUMN  3  ROWS  1 THRU  6  -----
ROW
1)  2.5890D-01  6.1310D-17  -9.4181D-01  5.3178D-17  1.0785D-01  -1.5260D-16
0COLUMN  4  ROWS  1 THRU  6  -----
ROW
1)  2.6658D-18  -9.3829D-03  7.6269D-17  9.9994D-01  1.2649D-16  -3.1716D-11
0COLUMN  5  ROWS  1 THRU  6  -----
ROW
1)  9.0713D-19  -6.4163D-11  2.0926D-17  3.1668D-11  -1.5600D-15  -1.0000D+00
0COLUMN  6  ROWS  1 THRU  6  -----
ROW
1)  1.4140D-02  -7.6485D-17  2.3385D-02  1.2233D-16  -9.9962D-01  1.5615D-15
  
```

Case Study - Post Test Evaluation, Summary

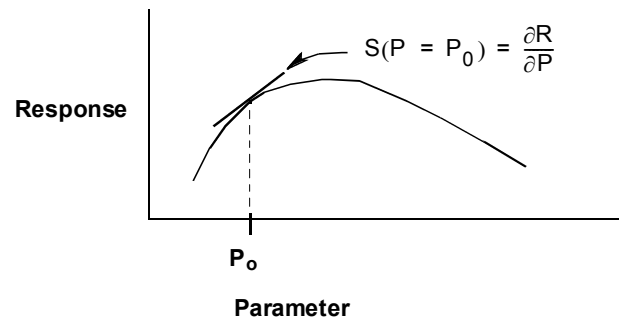
- The MAC/XORTHO matrices shows good agreement in mode shapes. (Off diagonal are small compared to the diagonal terms.)
- Mode 5 (test) is Mode 6 (analysis) and vice versa. This is shown by large off-diagonal terms in 5,6 and 6,5.
- Some effort is required to reformat test data for DMIG input.
- For a large number of modes, additional DMAP may be written to simplify the cross-correlation output.
- DMAP alters for pre test-analysis and post test-analysis MAC and cross-orthogonality, premaca.alt and postmaca.alt, is on the MSC Nastran delivery. There are also readme files to describe what the ALTERs do.

Model Refinement

- **There are three ways to update an MSC Nastran model to match test data:**
 1. Using brute force
 2. Using the sensitivity matrix
 3. Using design optimization
- **All methods update MSC Nastran model parameters such as 'I' and 'A' for BARs and 't' for QUADs. Base flexibility can also be a parameter if it was explicitly modeled (such as with ELASs).**
- **Brute force:**
 - Make model changes based on inspection of the results and make an educated “guess” as to the type of changes necessary.
 - After the model is updated, another analysis is made and the MSC Nastran results are again compared to test data to see if the match is better.
 - If the match is not better, then make more changes and repeat.

Model Refinement (cont.)

- **Sensitivity matrix:**
- **Sensitivity is the gradient of response with respect to model parameter. “Responses” are MSC Nastran-computed results, and “parameters” are property values.**



- **Each term in the sensitivity matrix S is given by $S_{ij} = \partial R_i / \partial P_j$**
- **where R_i is the i -th response and P_j is the j -th parameter.**
- **The greater the magnitude S_{ij} , the greater the response sensitivity.**
- **SOL 200 in MSC Nastran computes the sensitivity matrix.**

Model Refinement (cont.)

- **S** can be used by inspection to indicate which parameters need to be changed to change the response.
- **S** can also be output and used in a least-squares sense to minimize the difference between test and analysis as follows:

$$P_n = P_o + (S^T S)^{-1} S^T (R_t - R_a)$$

where P_n = updated parameters for test-analysis match

P_o = parameters from original model

S = sensitivity matrix

R_t = test response

R_a = analytical results

Model Refinement (cont.)

- **Design optimization:**

- Implemented in SOL 200
- Want to minimize the difference between test and analysis but with a model that changes least from the baseline model
- Use DEQATN to write an objective function that is the weighted difference between test data and analytical results.

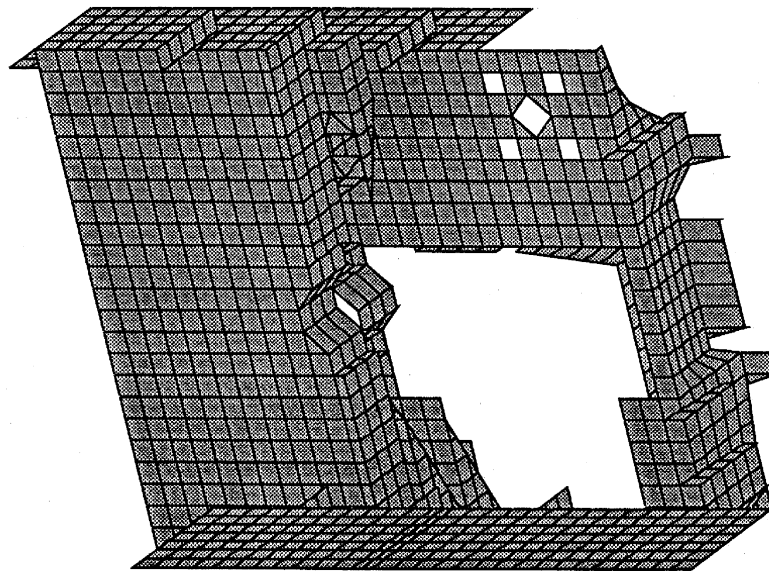
$$E = \text{wt} * \sum_{\substack{\text{test data} \\ i=1}} \text{WR}_i * (\text{RT}_i - \text{RA}_i)^2 + \text{wp} * \sum_{\substack{\text{model params} \\ j=1}} \text{WP}_j * (\text{PF}_j - \text{PO}_j)^2$$

where	RT _i	=	test data
	PF	=	final parameters
	RA _i	=	analysis results
	PO _j	=	original parameters
	WR _i	=	test weighting
	WP _j	=	parameter weighting
	wt	=	weighting for test as a whole
	wp	=	weighting for model parameters as a whole

- **SOL 200 minimizes the objective (E) subject to design constraints.**

Model Refinement (cont.)

- **Example:**
 - Disk drive enclosure with 1406 grid points, 1354 plate elements, four design variables (plate thicknesses), and four measured flexible modes
 - Minimize the difference between computed and measured resonant frequencies.



Model Refinement (cont.)

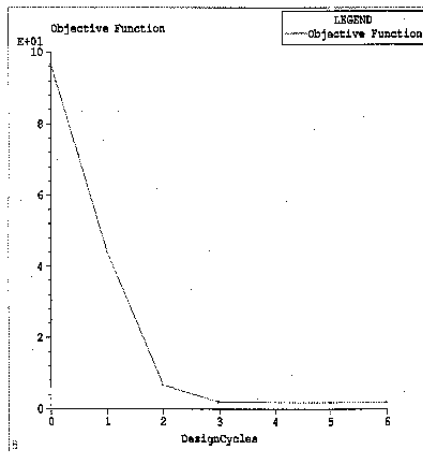
- **Example (Cont.)**

- Test data is weighted 100 times more than the baseline model parameters.
- Put upper and lower bounds on allowable plate thicknesses.
- Five iterations to converge
- Results:

Frequencies (Hz)		
Baseline	Test	Final
234	346	346
892	1307	1295
1165	1567	1460
1267	1678	1695

Plate Thicknesses (in.)			
Baseline	Lower Bound	Upper Bound	Final
.08	.05	.125	.125
.12	.05	.20	.075
.10	.05	.15	.084
.20	.10	.30	.227

Model Refinement (cont.)



Error versus Design Cycle

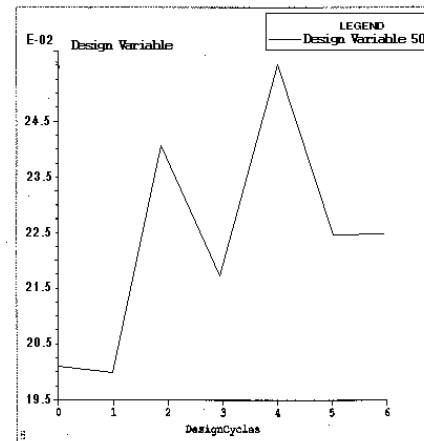
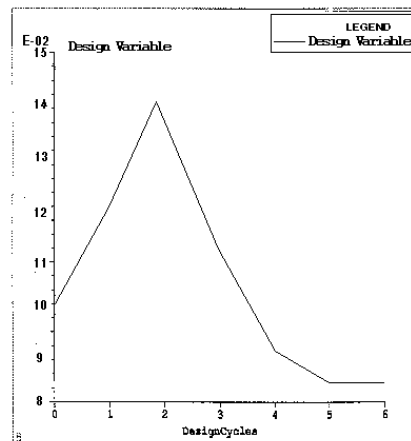
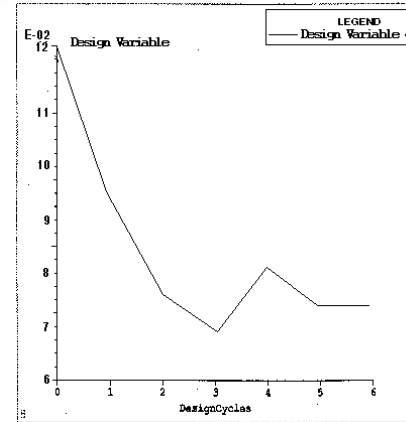
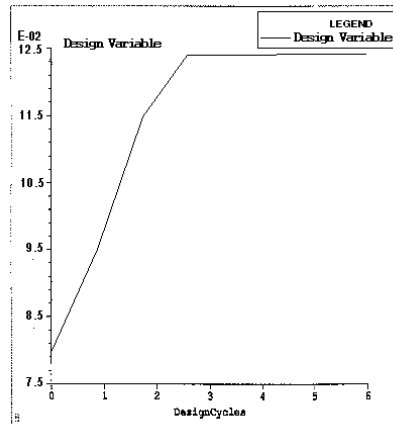


Plate Thickness versus Design Cycle

Model Refinement (cont.)

- Can also use SOL 200 to match frequency response test data.
- Can match static results as well.
- For best results, use as much test data as feasible (including the weight of the structure).
- Also, limit design variables to only the parameters that are truly uncertain.

References

Blakely, K. and Dobbs, M., “Integrated System Identification: The Union of Testing and Analysis,” *Proceedings First International Modal Analysis Conference*, November 1982.

Kientzy, D., Richardson, M. and Blakely, K., “Using Finite Element Data to Set Up Modal Tests,” *Sound and Vibration*, June 1989.

Blakely, K., “Updating MSC.Nastran Models to Match Test Data,” *Proceedings MSC World Users’ Conference*, March 1991.

Blakely, K. and Rose, T., “Cross-Orthogonality Calculations for Pre-Test Planning and Model Verification,” *Proceedings MSC World Users’ Conference*, May 1993.

Blakely, K., “Matching Frequency Response Test Data with MSC.Nastran,” *Proceedings MSC World Users’ Conference*, June 1994.

WORKSHOP 4

Perform an ASET selection and Compare with Test Results

MSC ProCOR

- MSC ProCOR is a graphical interface within MSC.Patran to correlate either:
 - Two different analytical models
 - An analytical model with corresponding test data
- MSC ProCOR modifies, marries, and exploits existing open architecture technology within MSC Patran and MSC Nastran to provide a comprehensive suite of correlation tools using:
 - MSC Nastran: DMAP Alters
 - Used as a mechanism to alter the solution sequence and perform additional calculations in support of correlation for modal analysis using MSC Nastran Solution 103.
 - MSC Patran: PCL
 - The mechanism to graphically interface with the FE model for analysis set-up, and post processing correlation exercises.
- MSC ProCOR isn't covered in this class, as NAS102b is a Nastran-only class. See the MSC ProCOR User Guide with the MSC Patran documentation for further details. This is only mentioned here since a lot of students want something more automated and graphical than what is presented in this chapter.

SECTION 9

FRF, FBA, & TPA

Frequency Response Function

- **Frequency Response was covered in the NAS102A seminar**
- **Additional features and applications are covered in this seminar**
 - FRF Frequency Response Function
 - FBA FRF Based Assembly
 - TPA Transfer Path Analysis
- **These features are widely used with NVH applications**

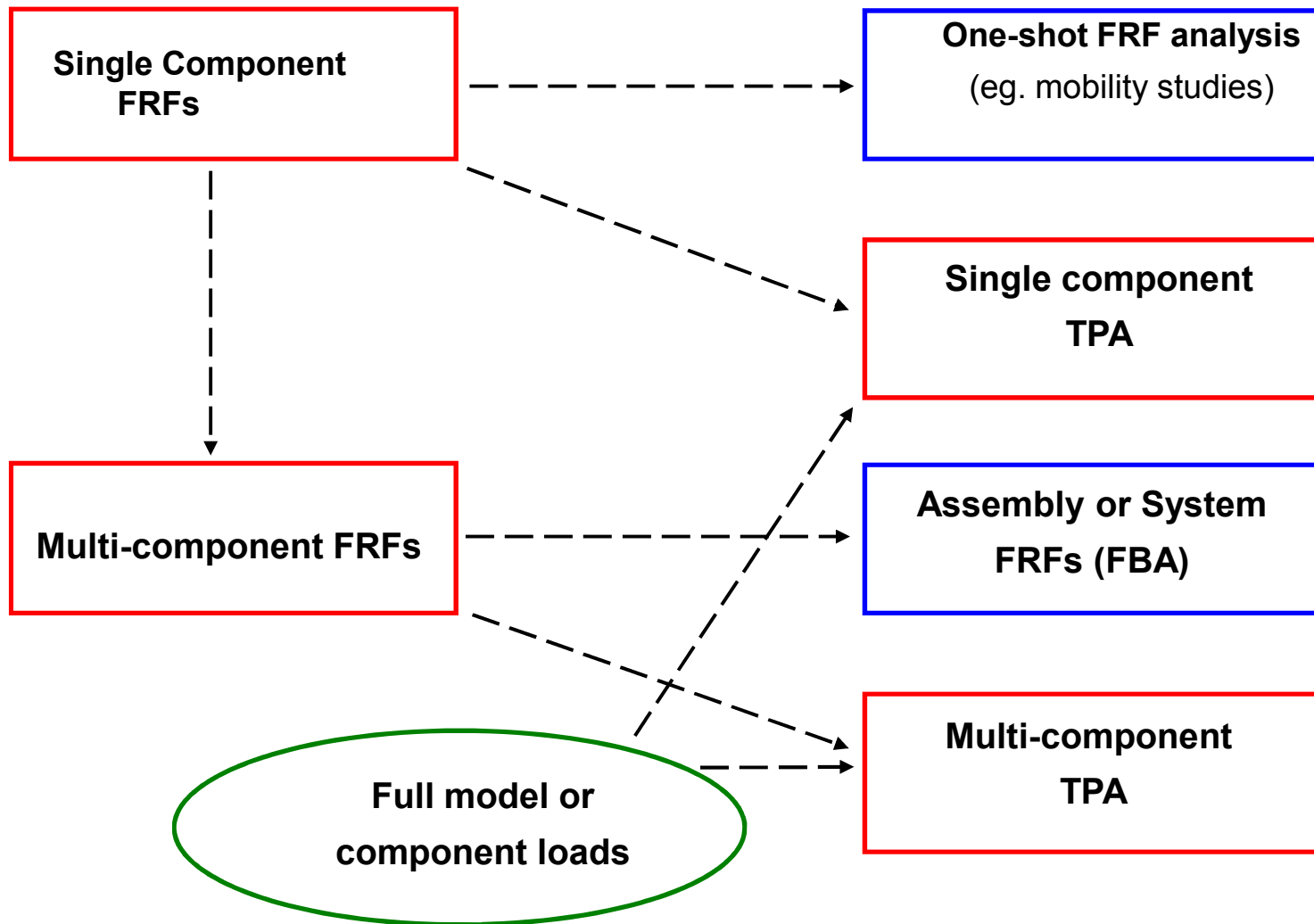
NVH Analysis Technique in MSC Nastran

- **Frequency Response Functions (FRFs) represent responses at desired DOFs of a component due to unit loads at specified DOFs of that component**
- **FRFs can be used to represent components**
- **FRFs of these components can then be employed for calculating the FRFs of assemblies of such components.**
 - This process is referred to as FBA (FRF Based Assembly).
- **FRFs of individual components or of an assembly of components can be used to obtain responses to user specified loads.**

NVH Analysis Technique in MSC Nastran (cont.)

- **FRF/FBA represents an alternative to external superelement (EXTSEOUT) based frequency response analysis.**
- **The technique also lends itself to tracking load paths or energy flows through a structure.**
 - This procedure is referred to as Transfer Path Analysis (TPA).
 - Can be used as a convenient way of replacing multiple subcases. In the auto industry, they frequently apply unit loads at different points in the structure and look at the response at different locations. This can take 100's of subcases. The FRF, FBA approach allows you to do that in one subcase by using the FRXITi entry.
- **TPA represents an alternative to mode participation studies**
 - Particularly well suited for vibration isolation, N&V mount studies and component design.

FRF / FBA / TPA Schematic



FRF Concept

- **Equation of Motion for Frequency Response Analysis:**

$$(-\omega^2[M] + i\omega[B] + [K])\{u\} = [Z]\{u\} = \{P\} \quad (1)$$

where $[Z]$ is the dynamic stiffness matrix

- **Solving for response displacement,**

$$\{u\} = [Z]^{-1}\{P\} = [H]\{P\} \quad (2)$$

where $[H]$ is the receptance matrix of the system. $[H]$ is also referred to as the FRF (Frequency Response Function) of the system.

- **Note that the FRF of a system is a function of the forcing frequency, ω ,**
 - Different FRF for the system for each forcing frequency.

FRF Concept (cont.)

- **FRF matrix at any forcing frequency represents the response of the system due to unit loads at that frequency.**
 - Rows of the FRF matrix represent response DOFs
 - Columns represent DOFs where unit loads are applied.
- **In practice, the FRF matrix is computed by solving the system for a series of unit loads.**
- **General FRF matrix is a square matrix**
 - Assumes that responses are computed at all DOFs
 - Unit loads are applied at all DOFs.
- **In most practical cases:**
 - Responses are computed at only a subset of the total points in a system
 - Loads are applied to only a small subset of the total DOFs in a system.
 - Actual FRF matrix to be considered for analysis purposes is rectangular
 - Number of rows is equal to the number of actual response DOFs
 - Number of columns is equal to the number of actual excitation DOFs.

FRF Concept (cont.)

Definition of FRFs

Response Function R	Standard FRF: $R/F(\text{orce})$	Inverse FRF: $F(\text{orce})/R$
Displacement	Receptance Admittance Dynamic Compliance Dynamic Flexibility	Dynamic Stiffness
Velocity	Mobility	Mechanical Impedance
Acceleration	Accelerance Inertance	Apparent Mass

(Source: “*Modal Testing: Theory, Practice and Application*,” by D. J. Ewins)

FBA Concept

- **FRFs of an assembly of components can be computed from the FRFs of the individual components by employing conditions of equilibrium and compatibility at the connected DOFs.**
 - This process is referred as FRF Based Assembly (FBA) and sometimes referred as FRF Based Substructuring (FBS).
- **Displacement Compatibility Condition**
 - Response displacement at a connected DOF must be the same for all components connected at that DOF.
- **Force Equilibrium Condition**
 - The forces at a connected DOF must be the sum of the forces from all components connected at that DOF.
- **Using the above two conditions, the FRFs of an assembly of components are computed in MSC Nastran solely from the FRFs of its constituent components and the connection information between and among these components. No other information is needed for the FBA process.**
- **The FBA process assumes**
 - FRFs of all FRF components are generated at the same forcing frequencies
 - These are also the forcing frequencies at which the FBA process is performed.

TPA Concept

- If the load $\{P\}$ is specified, then the partial response displacements $\{u\}_j$ due to the load P_j on the j -th DOF is given by

$$\{u\}_j = P_j \{H\}_j \quad (3)$$

where $\{H\}_j$ is the j -th column of the FRF matrix $[H]$

- The total response displacements $\{u\}$ due to the load $\{P\}$ is given by

$$\{u\} = \sum_{j=1}^n \{u\}_j = \sum_{j=1}^n P_j \{H\}_j = [H]\{P\} \quad (4)$$

- The TPA of a component involves the examination of the partial and total response displacements defined above due to the load $\{P\}$.
- The TPA of an assembly of components involves the examination of the partial and total response displacements for all of the constituent components as well as the computation and examination of the forces between and among the components due to specified loads.

Comparison Between External Superelement (EXTSEOUT) and FRF Based Approaches for Frequency Response Analysis

	EXTSEOUT Based Approach	FRF Based Approach
Component processing trigger	EXTSEOUT Case Control command	FRF Case Control command
Component processing procedure	Component mode synthesis (CMS) is a REQUIREMENT	Component mode synthesis (CMS) may be performed, but it is NOT a requirement
Component processing results	Boundary matrices	FRFs (via a complete SOL 108 or SOL 111 execution)
Residual	General structure	Limited connections
Assembly	Assembly of component boundary matrices (including those of the residual, if any)	Assembly of the component FRFs
Solution	Frequency response analysis of the assembled system via SOL108 or SOL 111 (preceded by computation of system modes for SOL 111)	Computation of the FRFs of the assembly by the FBA process
Data recovery	For the assembled configuration and for specified output requests of upstream components	For the assembled configuration and for specified output requests of upstream components
Analysis goals	Mode participation studies and optimization	Load path analysis (TPA) and optimization (future)

Advantages of FRF Based Approach

- **Provides an alternative to external superelement (EXTSEOUT) based approach for understanding system dynamic response**
- **Can point to attachments through which energy travels (TPA)**
- **Facilitates investigation of the dynamics of coupling between components to understand how to improve isolation**

Advantages of FRF Based Approach (cont.)

- **An attractive option for repetitive frequency response analyses involving multiple components**
- **Particularly for cases where the frequency response calculations represents a significant portion of the total time taken for the calculations**

Advantages of FRF Based Approach (cont.)

- **Can be used to attach FE based models with test based models**
 - FE models of certain components may have FRFs computed analytically
 - Test based models of other components may have FRFs determined by tests
 - FRFs of the above components can be combined together to yield FRFs of their assemblies

FRF/FBA/TPA Implementation

- **Provides an elegant way of computing FRFs of a component**
 - **Useful also for stand-alone FRF studies like performing mobility analysis**
 - **Avoid the cumbersome set up of one subcase per unit load**
- **Provides a convenient way of calculating the FRFs of an assembly of components from the FRFs of the individual components (FBA)**
- **FRF and FBA are supported for**
 - **Direct Frequency Response Analysis (SOL 108)**
 - **Modal Frequency Response Analysis (SOL 111).**

FRF/FBA Usage Procedure

- **Two steps procedure:**
- **Step 1 – FRF Generation**
 - **Generate the FRFs of an one shot configuration or of a component via the use of the FRF Case Control command in SOL 108 or SOL 111**
 - If this involves an one shot configuration, Step 2 is not needed. This is useful for stand-alone FRF studies like performing mobility analysis.
 - Otherwise, the user should define the points at which this FRF component is to be connected with other FRF components in a subsequent FBA process
 - FRFs and other information for the component will be saved on the medium specified by the FRF command (database or OP2 file)
 - **Generate assembly punch (.asm) file containing single (new) FRFCOMP Bulk Data entry for subsequent use in Step 2.**
 - **Unit loads - Simple specification via new FRFXIT/FRFXIT1 Bulk Data entries**
 - **A much better alternative to the specification of unit loads via multiple subcases with multiple DLOADs pointing to multiple RLOADi Bulk Data entries**
 - **User defined loads - Specification via the standard DLOAD Case Control command pointing to DLOAD/RLOADi Bulk Data entries**

FRF/FBA Usage Procedure (cont.)

- **Step 1 – FRF Generation (cont.)**
 - **FRFs are generated for the following points:**
 - **All points specified via DISP, VELO or ACCE output requests**
 - **All grid points associated with PLOTTEL entries**
 - **All points associated with elements for which STRESS/FORCE requests are specified**
 - **All points where unit loads and/or user specified loads are applied**
 - **All connection points**

FRF/FBA Usage Procedure (cont.)

- **Step 2 – FBA Process**
 - **Combine the FRFs of the components from Step 1 and compute the FRFs of an assembly of these components using the FBA process via the use of the FRF Case Control command in SOL 108 or SOL 111**
 - **FRF components to be combined in the FBA process are specified via FRFCOMP Bulk Data entries**
 - **These entries are in the .asm files generated by the FRF generation runs and to be included in the FBA job**
 - **Connections between and among the FRF components are normally determined automatically for grid points**
 - **FRFCONN entries may be employed to specify connections between scalar points and also explicit connections between coincident grid points.**

FRF/FBA Usage Procedure (cont.)

- **Step 2 – FBA Process (cont)**
 - **Connections between and among the FRF components are normally regarded as rigid connections**
 - **Flexible connections between coincident grid point DOFs may be specified via FRFFLEX Bulk Data entries by defining elastic, damping and damping coefficient values.**
 - These values may be constant at all frequencies or may be specified as frequency dependent values.
 - **Releases for grid point connection DOFs may be specified via FRFRELS Bulk Data entries**

FRF/FBA Usage Procedure (cont.)

- **Step 2 – FBA Process (cont'd)**
 - **Single-point constraints for connection DOFs may be specified via FRFSPC1 Bulk Data entries**
 - These entries are selected by the SPC Case Control command
 - **Unit loads – Already defined by the constituent FRF components**
 - **User defined loads - Specification via the FBALOAD, FBADLAY and FBAPHAS Bulk Data entries**
 - These entries essentially define complex scale factors by which the unit load responses will be multiplied in order to get responses to user defined loads
 - These entries are selected by the RLOAD1/RLOAD2 Bulk Data entries
 - These entries are converted internally by the program to equivalent DAREA, DELAY and DPHASE Bulk Data entries, respectively

FRF/FBA Usage Procedure (cont.)

- **Alternate Step 2 – FRF Generation of a Component and FBA Process in Same Execution**
 - **Generate the FRFs of a component via the use of the FRF Case Control command in SOL 108 or SOL 111**
 - **In the same run, combine these FRFs with the FRFs of other components from Step 1 to compute the FRFs of an assembly of these components using the FBA process**
 - **This scenario is particularly useful for design iterations or design alternatives involving the last FRF component.**

FRF/FBA Usage Procedure (cont.)

- **Results for user defined loads from both FRF and FBA jobs**
 - **If the user specifies a dynamic load in a FRF / FBA job (SOL 108 or SOL 111) via a DLOAD Case Control request and the load involves loads on N DOFs in the model,**
 - Nastran automatically generates results for (N+1) load cases.
 - First N of these load cases represent loads on each of the N DOFs separately
 - (N+1)th load case represents the total applied load.
 - **Option available to output only the (N+1)th load case (total load)**

FRF/FBA Usage Procedure (cont.)

- **Results for user defined loads from both FRF and FBA jobs (cont.)**
 - **For standard SOL 108 / SOL 111 jobs**
 - Output is generated only for the total applied load, corresponding to the $(N+1)^{\text{th}}$ load case mentioned above.
 - **The output for user defined loads from FRF / FBA jobs described above is perfectly suited for TPA studies**
 - It allows for the examination of the effects on the response of the system due to loads on individual DOFs as well as due to the total applied load.

Case Control and Bulk Data for FRF/FBA Usage

- **Case Control for FRF/FBA Jobs**
 - FRF (Analysis Type, Component ID and Name, Output Medium, etc.)
- **Bulk Data for FRF Generation Jobs**
 - Unit Load Specification
 - FRFXIT
 - FRFXIT1
- **Bulk Data for FBA Jobs**
 - Component Specification for FBA Process
 - FRFCOMP
 - Connectivity Data
 - FRFCONN
 - FRFFLEX
 - FRFRELS
 - FRFSPC1
 - User Load Specification
 - FBADLAY
 - FBALOAD
 - FBAPHAS

FRF Case Control Command

Format:

$$\begin{aligned}
 & \text{FRF} \left[\left(\left[\begin{array}{c} \text{GEN} \\ \text{ASM} \\ \text{GENASM} \end{array} \right] \right) [\text{COMPID} = \text{cmpid}] [\text{COMPNAME} = \text{cmpname}] \right. \\
 & \quad \left. [\text{CONNPTS} = \text{setid}] \left[\text{XITOUT} = \left\{ \begin{array}{c} \text{UNIT} \\ \text{UNITALL} \\ \text{USER} \\ \text{USERTOTL} \end{array} \right\} \right] \left[\text{ASMOUT} = \left\{ \begin{array}{c} \text{COMP} \\ \text{ALL} \\ \text{ASSEMBLY} \\ \text{CONNINFO} \\ n \\ \text{cname} \end{array} \right\} \right] \left[\text{LOADLBL} = \left\{ \begin{array}{c} \text{STD} \\ \text{ALT} \\ \text{ALT X} \end{array} \right\} \right] \\
 & \quad \left. \left\{ \begin{array}{l} \text{ICFGEN} = \left\{ \begin{array}{c} \text{ALL} \\ n \\ \text{compname} \end{array} \right\} \\ \text{ICFUSE} = \left\{ \begin{array}{c} n \\ \text{compname} \end{array} \right\} \\ \text{ICFAUTO} = \left\{ \begin{array}{c} n \\ \text{compname} \end{array} \right\} \end{array} \right\} = \left\{ \begin{array}{c} \text{ICFDB} \\ \text{ICFOP2} = \{\text{icfunit}\} \end{array} \right\} \right.
 \end{aligned}$$

EXAMPLES

- **Some example usages are listed below**
 - Will explain functions in details later on

Examples:

FRF

FRF (COMPID = 10 COMPNAME = WING CONNPTS = 1000 LOADLBL = ALT)

FRF (COMPID = 20 COMPNAME = STRUT CONNPTS = 2000 OP2 = 25)

FRF (COMPID = 30 COMPNAME = BODY CONNPTS = 300 XITOUT = UNITALL)

FRF (ASM)

FRF (ASM LOADLBL = ALTX)

FRF (ASM ASMOUT = ALL)

FRF (GENASM COMPID = 50 COMPNAME = SUSPENS CONNPTS = 200)

FRF (ASM ICFGEN = ALL)

FRF (ASM ICFUSE = 100)

FRF (ASM ICFGEN = 200 ICFOP2 = 33)

FRF (ASM ICFUSE = -1 ICFOP2 = 33)

FRF (ASM ICFAUTO = 100)

FRF (ASM ICFAUTO = -5 ICFOP2 = 31)

FRF Case Control Command (cont.)

Generation/Assembly Keyword

- **GEN** **Generates the FRFs for the specified component (default)**
- **ASM** **Computes the FRFs of an assembly from the components generated by the generation run(s)**
- **GENASM** **Computes the FRF for a specified component and assemble them in the same run with the FRFs from other generation run(s).**

FRF Case Control Command (cont.)

Component ID/Name Keyword

- **COMPID = cmpid** ID number of the FRF component
- **COMPNAME = cmpname** Name of the FRF component

Connection Keyword

- **CONNPTS= SETID** *setid (integer > 0)* refers to the set that defines the points at which the FRF component specified by the **COMPID/COMPNAME** keywords is to be connected during a subsequent FRF based assembly (FBA) process.

FRF Case Control Command (cont.)

FRF Output Control Keyword

- **XITOUT=unit** Output the FRF results only for unit excitations that are specified explicitly via FRFXIT / FRFXIT1 Bulk Data entries or implicitly via the DLOAD Case Control request
- **XITOUT=unitall** Output FRF results for unit excitations specified explicitly via FRFXIT / FRFXIT1 Bulk Data entries or implicitly via the DLOAD Case Control command,

Plus output for unit excitations that are internally applied automatically at the connection points of the FRF component(s).

FRF Case Control Command (cont.)

FRF Output Control Keyword (cont.)

- **XITOUT=USER** Output the FRF results for the following excitations implied by the DLOAD Case Control request:
 - a.) A separate excitation for each individual DOF that has a nonzero load value specified for it (1 to Nth case)
 - b.) An excitation representing the total load (N+1)th case

For each subcase, the (N+1) excitations are identified in the form xxxxyyyy

xxxx = Subcase ID

yyyy = range from 1 through N

The (N+1) excitation uses xxxx9999

FRF Case Control Command (cont.)

FRF Output Control Keyword (cont.)

- **XITOUT=USERTOT** **Output the FRF results for the single excitation representing the total load implied by the DLOAD Case Control request. This corresponds to the (N+1)th excitation with the XITOUT=USER option**

FRF Case Control Command (cont.)

ASMOUT Keyword

- **FRF (...ASMOUT= COMP)**

In the FBA run, output only the individual FRF components (default)

- **FRF (...ASMOUT=ASSEMBLY)**

Generate output from the FBA process only for the assembled FRF configuration. This is the same as the ASMOUT=0 option

- **FRF (...ASMOUT=ALL)**

In the FBA run, output the individual FRF components plus the assembled configuration

FRF Case Control Command (cont.)

ASMOUT Keyword (cont.)

- **FRF (...ASMOUT=CONNINFO)**

Terminate the FBA run after generating the FRF component connection information without performing any further FRF assembly operations (like a check run)

- **FRF (...ASMOUT=cname)**

Generate output from the FBA process only for the single FRF component with the name of cname

FRF Case Control Command (cont.)

ASMOUT Keyword (cont.)

- **FRF (...ASMOUT=n)**

n is an integer with the following definition

n=0

Generate output from the FBA process only for the assembled FRF configuration. This is the same as ASMOUT=ASSEMBLY

n > 0

Generate output from the FBA process only for those FRF components defined by the SET ID n

n < 0

Generate output from the FBA process only for the single FRF components defined by the |n|

FRF Case Control Command (cont.)

LOADLBL Keyword

- **This keyword in the FRF command is used to control the load labels in the output of the FRF generation and FBA assembly runs**
- **The label has the following options and meaning:**

FRF (...LOADLBL = STD)

- The load labels identify the grid (or scalar) point and its component where the load is applied (default)

FRF Case Control Command (cont.)

LOADLBL Keyword (cont.)

FRF (...LOADLBL = ALT)

- The load labels identify the grid (or scalar) point and its component where the load is applied.
- It uses the following convention:

GGGGGGGG + C

where GGGGGGGG is the grid (or scalar) ID and C indicates the component where the load is applied

X – grid point component 1 or scalar point

Y – grid point component 2

Z – grid point component 3

RX – grid point component 4

RY – grid point component 5

RZ – grid point component 6

FRF Case Control Command (cont.)

LOADLBL Keyword (cont.)

- FRF (...LOADLBL = ALTX)
 - Same meaning as ALT plus
 - Identify where the applied load is an unit load or user defined load

FRF Case Control Command (cont.)

ICFGEN Keyword

- **FRF (.....ICFGEN = ALL)**
 - Generate ICF information in the FBA process for all of the FRF components of the assembly
- **FRF (.....ICFGEN = n)**
 - $n > 0$ Generate ICF information in the FBA process for the FRF components defined by the SET=n
 - $n < 0$ Generate ICF information in the FBA process for component ID |n|
- **FRF (...ICFGEN = compname)**
 - Generate ICF for component with name of compname

FRF Case Control Command (cont.)

ICFUSE Keyword

- **FRF (.....ICFUSE = ALL)**
 - Use ICF information in the FBA process for all of the FRF components of the assembly
- **FRF (.....ICFUSE = n)**
 - $n > 0$ Use ICF information in the FBA process for the FRF components defined by the SET=n
 - $n < 0$ Use ICF information in the FBA process for component ID |n|
- **FRF (...ICFUSE = compname)**
 - Use ICF for component with name of compname

FRF Case Control Command (cont.)

ICFAUTO Keyword

- **FRF (.....ICFAUTO = n)**
 - $n > 0$ Generate and use ICF information in the FBA process for the FRF components defined by the SET=n
 - $n < 0$ Generate and use ICF information in the FBA process for component ID |n|
- **FRF (...ICFUSE = compname)**
 - Generate and use ICF for component with name of compname

FRF Case Control Command (cont.)

ICF Storage Method

- **FRF (...ICFDB)**
 - ICF information is to be stored on the database. This is the default.
- **FRF (...ICFOP2= xx)**
 - ICF information is to be stored on the database where xx is the unit number, e.g., 41)

FRF Related Bulk Data Entries

- **Unit Load specification**
 - FRFXIT
 - FRFXIT1
- **Component Specification for FBA Process**
 - FRFCOMP
- **Connectivity Data**
 - FRFCONN
 - FRFFLEX
 - FRFRELS
 - FRFSPC1
- **User Load Specification**
 - FBADLAY
 - FBALOAD
 - FBAPHAS

Unit Load Specification

- **FRFXIT** Specifies DOF where unit load is to be for FRF

Format:

1	2	3	4	5	6	7	8	9	10
FRFXIT	PNTID	C	LABEL						

Example:

FRFXIT	10	3	UNIT LOAD AT LEFT CORNER				

Field	Contents
PNTID	Grid or scalar point identification number. (Integer > 0)
C	A single component number. (Integer 0 or blank for a scalar point; any integer between 1 and 6 for a grid point.)
LABEL	A string comprising no more than 48 characters (fields 4 through 9) that will be used in the label portion of the FRF output to identify this unit load specification. See Remarks 1. and 2.

Unit Load Specification (cont.)

- **FRFXIT1** Specifies DOFs where unit loads are to be for FRF

Format:

1	2	3	4	5	6	7	8	9	10
FRFXIT1	C	PNTID1	PNTID2	PNTID3	PNTID4	PNTID5	PNTID6	PNTID7	

Example:

FRFXIT1	123	10	20	30	40				

Alternate Format and Example:

1	2	3	4	5	6	7	8	9	10
FRFXIT1	C	PNTID1	THRU	PNTID2					

FRFXIT1	123	5	THRU	15					

Field	Contents
C	Component numbers. (Any unique combination of integers 1 through 6 with no embedded blanks for grid points; integer 0 or blank for scalar points.)
PNTIDi	Grid or scalar point identification numbers. See Remark 1. (Integer > 0)

Remarks:

1. Points in the THRU range need not all exist.
2. The FRFXIT Bulk Data entry and the DLOAD Case Control request provide alternate means of unit load specification for FRF generation.
3. Redundant unit load specifications are ignored.

Component Specification for FBA Process

- **FRFCOMP** Specifies FRF component to be assembled

Format

1	2	3	4	5	6	7	8	9	10
FRFCOMP	COMPID	COMPNAME	MEDIUM	UNITNO	LSCALFAC	FSCALFAC			

Examples:

FRFCOMP	10	LEFTWING	OP2	25					
FRFCOMP	20	FRAME	UF	30	1000.0				

- COMPID Component ID
- COMPNAME Component Name
- MEDIUM DB (database)
 OP2 (output2 file)
 UF (universal file)

Component Specification for FBA Process (cont.)

- UNITNO - Fortran unit number for op2 and UF files
- LSCALEFAC - Length scale factor for the UF option
- FSCALEFAC - Force scale factor for the UF option

FRF Connectivity Data

- **FRFCONN** Defines explicit connection for FRF based assembly process (FBA)

Format:

1	2	3	4	5	6	7	8	9	10
FRFCONN	CONNID	COMPID1/ COMPNAM1	POINT1	COMPID2/ COMPNAM2	POINT2				

Examples:

FRFCONN	10	5	100	15	200				
FRFCONN	20	STRUT	25	WING	35				

Field	Contents
CONNID	Unique identification number of the FRFCONN entry. (Integer > 0)
COMPIDi	Identification number of an FRF component whose FRFs have been generated in a previous Nastran execution. See Remark 1. (Integer > 0)
COMPNAMi	Name of an FRF component whose FRFs have been generated in a previous Nastran execution. See Remark 1. (Up to 8 characters; no blank allowed)
POINTi	Grid or scalar point identification number. See Remarks 2. through 5. (Integer > 0)

FRF Connectivity Data (cont.)

- **FRFFLEX** Defines flexible properties (K, B, GE) for flexible connections between FRF components (default is rigid)

Format:

1	2	3	4	5	6	7	8	9	10
FRFFLEX	FLEXID	C	COMPID1/ COMPNAME1	POINT1	COMPID2/ COMPNAME2	POINT2	KVALUE/ KTABID	BVALUE/ BTABID	
	GEVALUE/ GETABID								

Examples:

FRFFLEX	10	1	100	15	200	25	100.0	150	
	0.02								

FRFFLEX	20	4	STRUT	120	WING	260	10	0.25	

FRF Connectivity Data (cont.)

- **FRFFLEX (cont.)**

Field	Contents
FLEXID	Unique identification number of the FRFFLEX entry. (Integer > 0)
C	A single component number. (Any integer between 1 and 6 for grid points; integer 0 or blank for scalar points.)
COMPIDi	Identification number of an FRF component whose FRFs have been generated in a previous Nastran execution. See Remark 1. (Integer > 0)
COMPNAMEi	Name of an FRF component whose FRFs have been generated in a previous Nastran execution. See Remark 1. (Up to 8 characters; no blank allowed)
POINTi	Grid or scalar point identification number. See Remark 2. (Integer > 0)
KVALUE	Elastic property value (force per unit displacement). See Remark 3. (Real > 0.0 or blank)
KTABID	Identification number of a TABLEDi entry that defines the elastic property value (force per unit displacement) as a function of frequency. See Remark 3. (Integer > 0 or blank)
BVALUE	Damping property value (force per unit velocity). See Remark 3. (Real > 0.0 or blank)
BTABID	Identification number of a TABLEDi entry that defines the damping property value (force per unit velocity) as a function of frequency. See Remark 3. (Integer > 0 or blank)
GEVALUE	Damping coefficient value. See Remarks 4., 5. and 6. (Real > 0.0 or blank)
GETABID	Identification number of a TABLEDi entry that defines the damping coefficient value as a function of frequency. See Remarks 4., 5. and 6. (Integer > 0 or blank)

FRF Connectivity Data (cont.)

- FRFRELS** Defines the DOFs of the FRF component connection grid points that are not to be connected in the FBA run

Format:

1	2	3	4	5	6	7	8	9	10
FRFRELS	SID	C	COMPID1/ COMPNAM1	GRIDPNT1	COMPID2/ COMPNAM2	GRIDPNT2	COMPID3/ COMPNAM3	GRIDPNT3	

Examples:

FRFRELS	100	45	10	15	BODY	20	FRAME	30	
FRFRELS	20	1	WING	25	NACELLE	35			

Field	Contents
SID	Identification number of the FRFRELS entry. (Integer > 0)
C	Component number(s). See Remark 1. (Any unique combination of the integers 1 through 6 with no embedded blanks.)
COMPIDi	Identification number of an FRF component whose FRFs have been generated in a previous Nastran execution. See Remark 2. (Integer > 0)
COMPNAMEi	Name of an FRF component whose FRFs have been generated in a previous Nastran execution. See Remark 2. (Up to 8 characters; no blank allowed)
GRIDPNTi	Grid point identification number. See Remarks 3. and 4. (Integer > 0)

FRF Connectivity Data (cont.)

- FRFSPC1** Defines single point constraints for FRF component connection points in FBA run

Format:

1	2	3	4	5	6	7	8	9	10
FRFSPC1	SID	C	COMPID1/ COMPNAM1	POINT1	COMPID2/ COMPNAM2	POINT2	COMPID3/ COMPNAM3	POINT3	

Examples:

FRFSPC1	100	4	10	12	ENGINE	23	FRAME	31	

FRFSPC1	20	1	STRUT	25	NACELLE	35			

Field	Contents
SID	Identification number of the single-point constraint set. See Remark 1. (Integer > 0)
C	Component number(s). (Any unique combination of the integers 1 through 6 with no embedded blanks for grid points; integer 0 or blank for scalar points.)
COMPIDi	Identification number of an FRF component whose FRFs have been generated in a previous Nastran execution. See Remark 2. (Integer > 0)
COMPNAMEi	Name of an FRF component whose FRFs have been generated in a previous Nastran execution. See Remark 2. (Up to 8 characters; no blank allowed)
POINTi	Grid or scalar point identification number. See Remark 3. (Integer > 0)

User Load Specification

- FBALOAD** Defines the scale factor for frequency-dependent loads in an FBA run

Format:

1	2	3	4	5	6	7	8	9	10
FBALOAD	SID	COMPID/ COMPNAME	PNTID	C	A				

Examples:

FBALOAD	10	WING	20	3	2.5				

FBALOAD	20	30	25	1	1.5				

Field	Contents
SID	Identification number. See Remark 1. (Integer > 0)
COMPID	Identification number of the FRF component whose FRFs have been generated in a previous Nastran execution. (Integer > 0)
COMPNAME	Optional name of the FRF component whose FRFs have been generated in a previous Nastran execution. (Up to 8 characters; no blank allowed).
PNTID	Grid or scalar point identification number. See Remark 3. (Integer > 0)
C	Component number. See Remark 3. (Integer 1 through 6 for grid point; blank or 0 for scalar point)
A	Scale factor. See Remark 4. (Real)

User Load Specification (cont.)

- FBADLAY** Defines the time delay for frequency-dependent loads in an FBA run

Format:

1	2	3	4	5	6	7	8	9	10
FBADLAY	SID	COMPID/ COMPNAME	PNTID	C	DELAY				

Examples:

FBADLAY	15	BODY	10	3	0.5				
FBADLAY	25	30	5	2	0.1				

Field	Contents
SID	Identification number. See Remark 1. (Integer > 0)
COMPID	Identification number of the FRF component whose FRFs have been generated in a previous Nastran execution. (Integer > 0)
COMPNAME	Name of the FRF component whose FRFs have been generated in a previous Nastran execution. (Up to 8 characters; no blank allowed)
PNTID	Grid or scalar point identification number. See Remark 3. (Integer > 0)
C	Component number. See Remark 3. (Integer 1 through 6 for grid point; blank or 0 for scalar point)
DELAY	Time delay term τ . See Remark 4. (Real)

User Load Specification (cont.)

- FBAPHAS** Defines the phase lead for frequency-dependent loads in an FBA run

Format:

1	2	3	4	5	6	7	8	9	10
FBAPHAS	SID	COMPID/ COMPNAME	PNTID	C	PHASE				

Examples:

FBAPHAS	25	FRAME	40	1	10.0				

FBAPHAS	30	50	10	2	5.0				

Field	Contents
SID	Identification number. See Remark 1. (Integer > 0)
COMPID	Identification number of the FRF component whose FRFs have been generated in a previous Nastran execution. (Integer > 0)
COMPNAME	Name of the FRF component whose FRFs have been generated in a previous Nastran execution. (Up to 8 characters; no blank allowed)
PNTIDi	Grid or scalar point identification numbers. See Remark 3. (Integer > 0)
C	Component number. See Remark 3. (Integers 1 through 6 for grid points; blank or 0 for scalar points.)
PHASE	Phase lead term θ . See Remark 4. (Real)

Relaxation of Connection Data

- Connection data (via FRFCONN and/or FRFFLEX/FRFRELS) is not required if there is only a single component
 - Warning message will be issued
- Convenient for load studies and Transfer Path Analysis (TPA) for a single FRF component

Support of Test Components

- In addition to analysis, FRF also supports components from testing
- Test component must be created in the Universal File (UF) format
- UF is grouped by the Universal Dataset Numbers (UDNs)
- Details for this format can be obtained from the following URL

<http://www.sdrl.uc.edu/universal-file-formats-for-modal-analysis-testing-1>

- For the FBA process, the bulk of the data is contained in the UDN 58

Support of Test Components (cont.)

- The universal file is attached by using the ASSIGN FMS statement
- An example is as follows:

```
ASSIGN UNVFILE='frfcomp1.unv' UNIT=25
```

- As with internal FRF component, you can attached multiple test components
 - One ASSIGN statement for each attached component

Support of Test Components (cont.)

- Following information regarding the FRF components are available in the FBA run
 - Component count
 - FRF component ID and name
 - How each FRF component is generated
 - From current MSC Nastran run
 - From previous MSC Nastran run(s)
 - From testing
 - Medium on which the FRF information resides
 - Database
 - Op2 with the unit #
 - Universal file with the unit #

EXAMPLES

Examples to Illustrate FRF/FBA Implementation in MSC Nastran

- **Four Examples**
 - Example 1. Flat plate – 8 components
 - Example 2. Automobile trim body and suspension system
 - Example 3. Flyswatter – 7 analytical comp, 1 test comp
 - Example 4. User Applied Loads
- **All of these models have been analyzed using the FRF/FBA implementation**
- **Results are compared using the FRF method versus the one shot method**

FRF/FBA Example 1

Rectangular Plate - Full Model

880		881		882		883		884		885		886		887		888
870	857	871	858	872	859	873	860	874	861	875	862	876	863	877	864	878
860	849	861	850	862	851	863	852	864	853	865	854	866	855	867	856	868
850	841	851	842	852	843	853	844	854	845	855	846	856	847	857	848	858
840	833	841	834	842	835	843	836	844	837	845	838	846	839	847	840	848
830	825	831	826	832	827	833	828	834	829	835	830	836	831	837	832	838
820	817	821	818	822	819	823	820	824	821	825	822	826	823	827	824	828
810	809	811	810	812	811	813	812	814	813	815	814	816	815	817	816	818
800	801	801	802	802	803	803	804	804	805	805	806	806	807	807	808	808

FRF/FBA Example 1 (cont.)

Rectangular Plate – 8 FRF Components

880		881		882		883		884
870	857	871	858	872	859	873	860	874
860	849	861	850	862	851	863	852	864

884		885		886		887		888
874	861	875	862	876	863	877	864	878
864	853	865	854	866	855	867	856	868

860		861		862		863		864
850	841	851	842	852	843	853	844	854
840	833	841	834	842	835	843	836	844

864		865		866		867		868
854	845	855	846	856	847	857	848	858
844	837	845	838	846	839	847	840	848

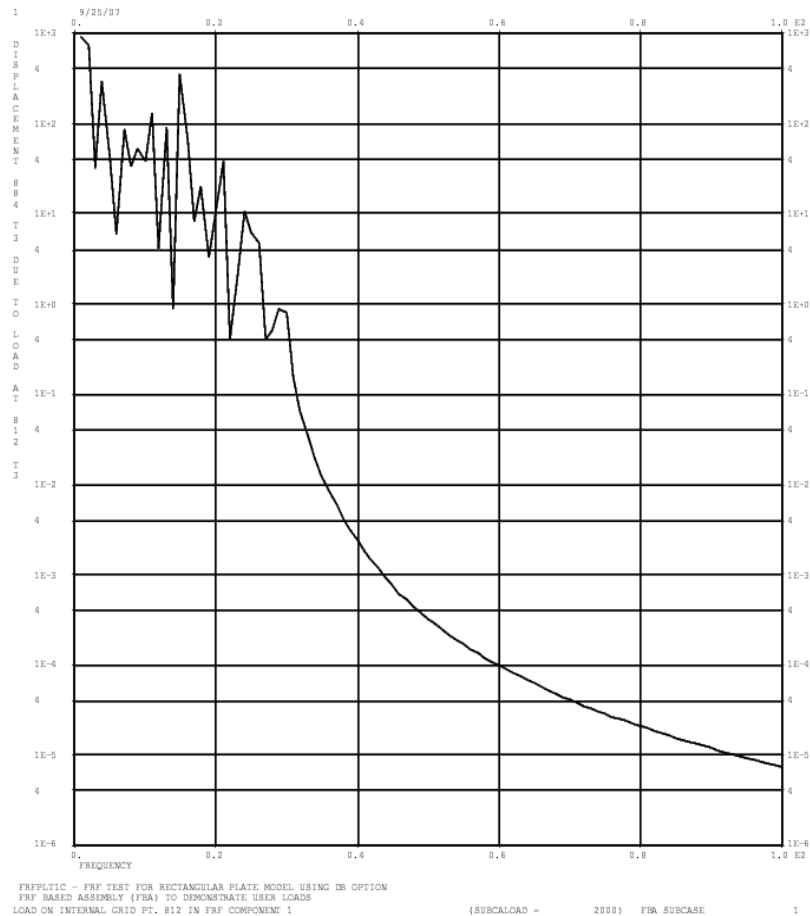
840		841		842		843		844
830	825	831	826	832	827	833	828	834
820	817	821	818	822	819	823	820	824

844		845		846		847		848
834	829	835	830	836	831	837	832	838
824	821	825	822	826	823	827	824	828

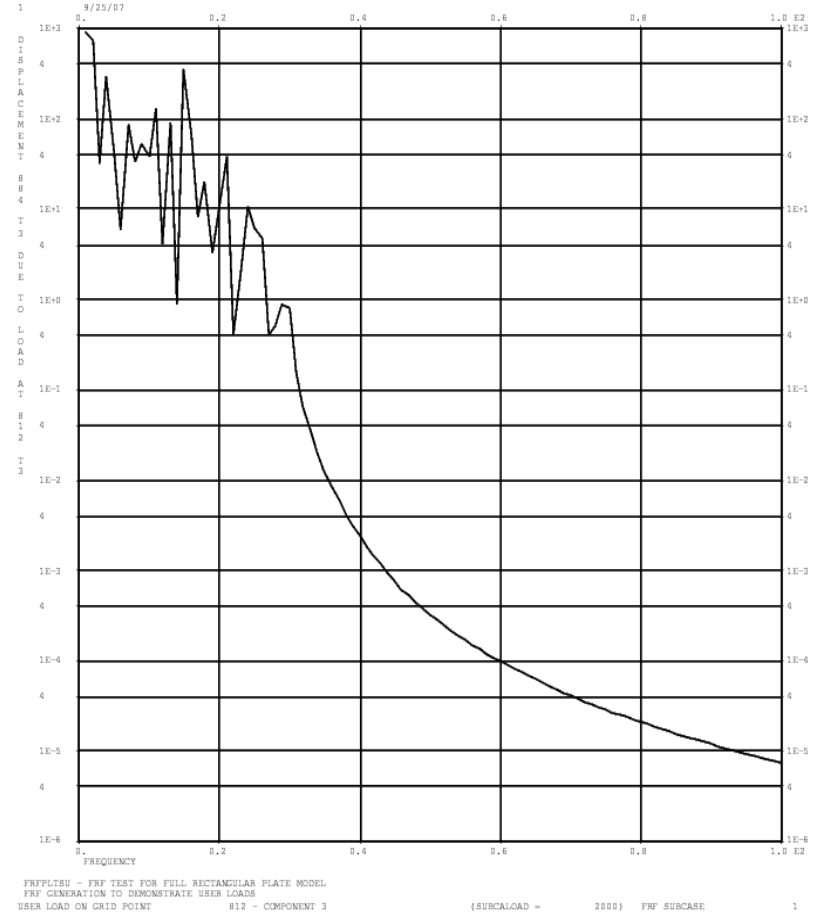
820		821		822		823		824
810	809	811	810	812	811	813	812	814
800	801	801	802	802	803	803	804	804

824		825		826		827		828
814	813	815	814	816	815	817	816	818
804	805	805	806	806	807	807	808	808

FRF/FBA Example 1 (cont.)
FRF/FBA Results vs Single Shot Results
Displacement at Grid 884(T3) Due to User Load at Grid 812(T3)
(All Connections Are Rigid)



FRF/FBA Results



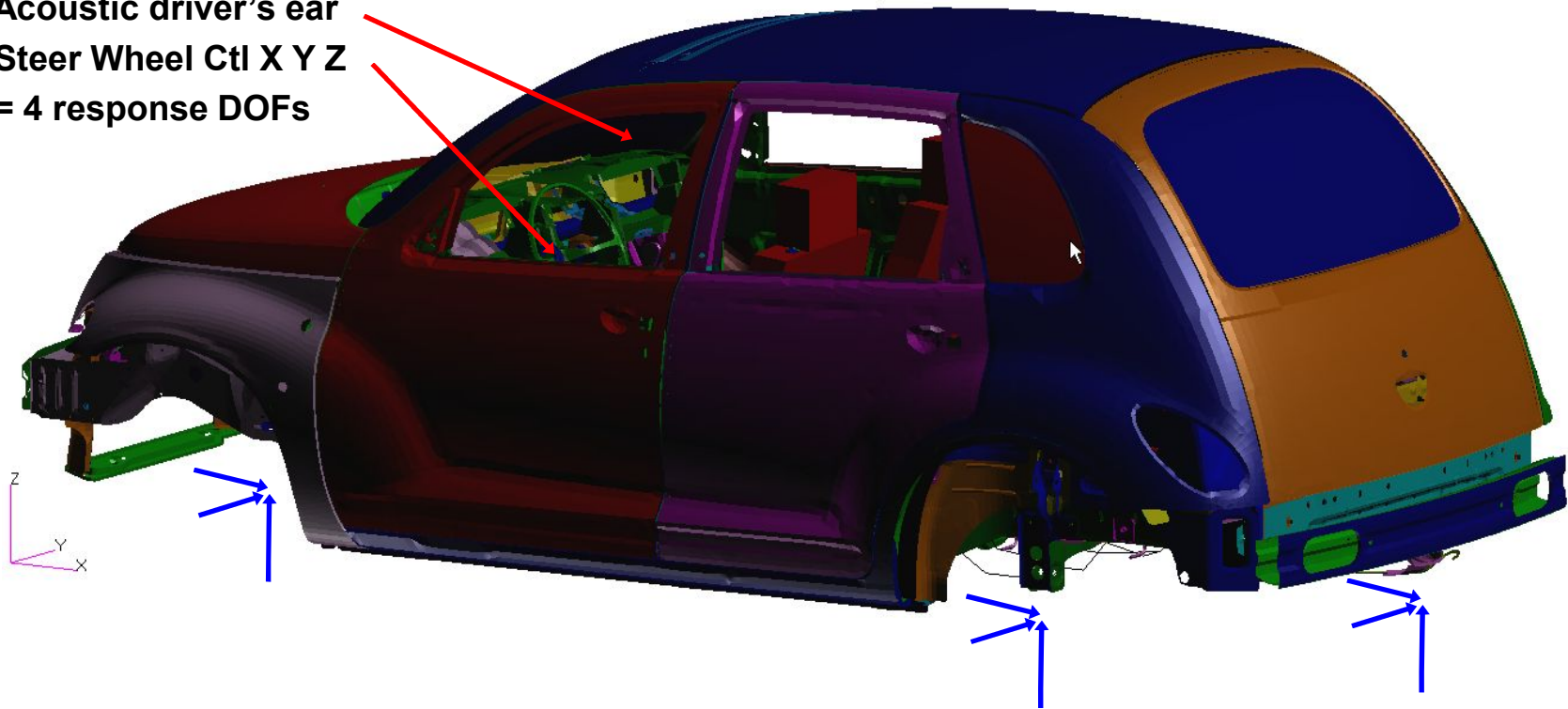
Single Shot Results

FRF/FBA Example 2

Detailed Trim Body (with Acoustic Cavity)

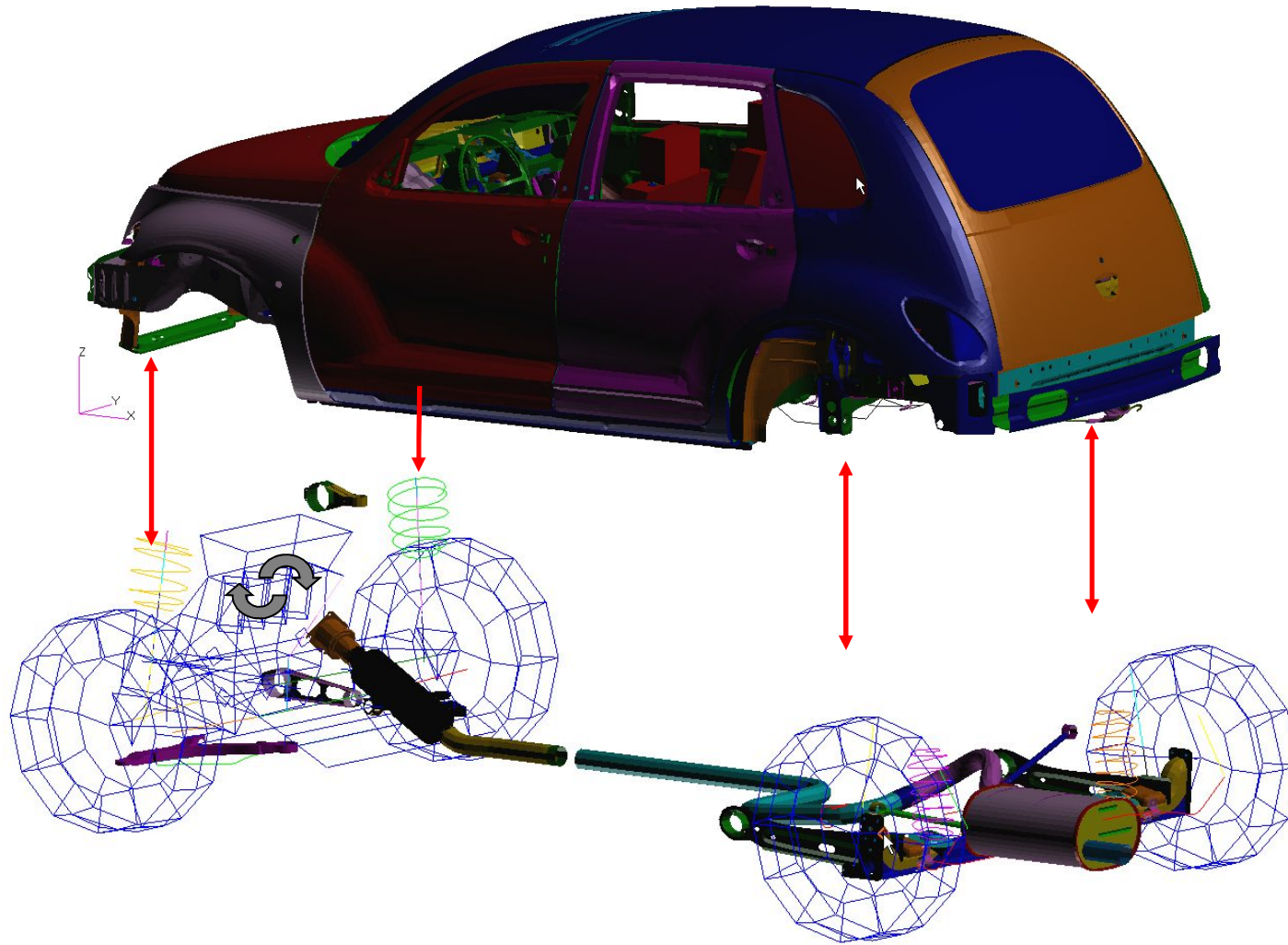
Responses

Acoustic driver's ear
Steer Wheel Ctl X Y Z
= 4 response DOFs

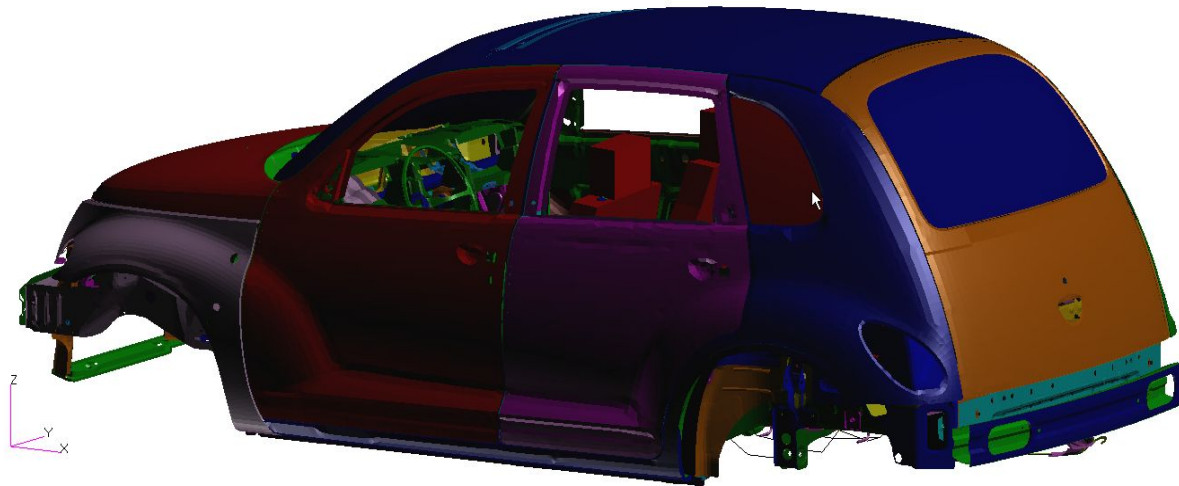


FRF/FBA Example 2 (cont.)

Detailed Trim Body + Suspension System & Idle Engine Torque Loads



FRF/FBA Example 2 (cont.) Detailed Trim Body FRF



2M DOFs

20 – 300 Hz FRF

3,300 struct modes to 450 Hz

150 fluid modes to 600 Hz

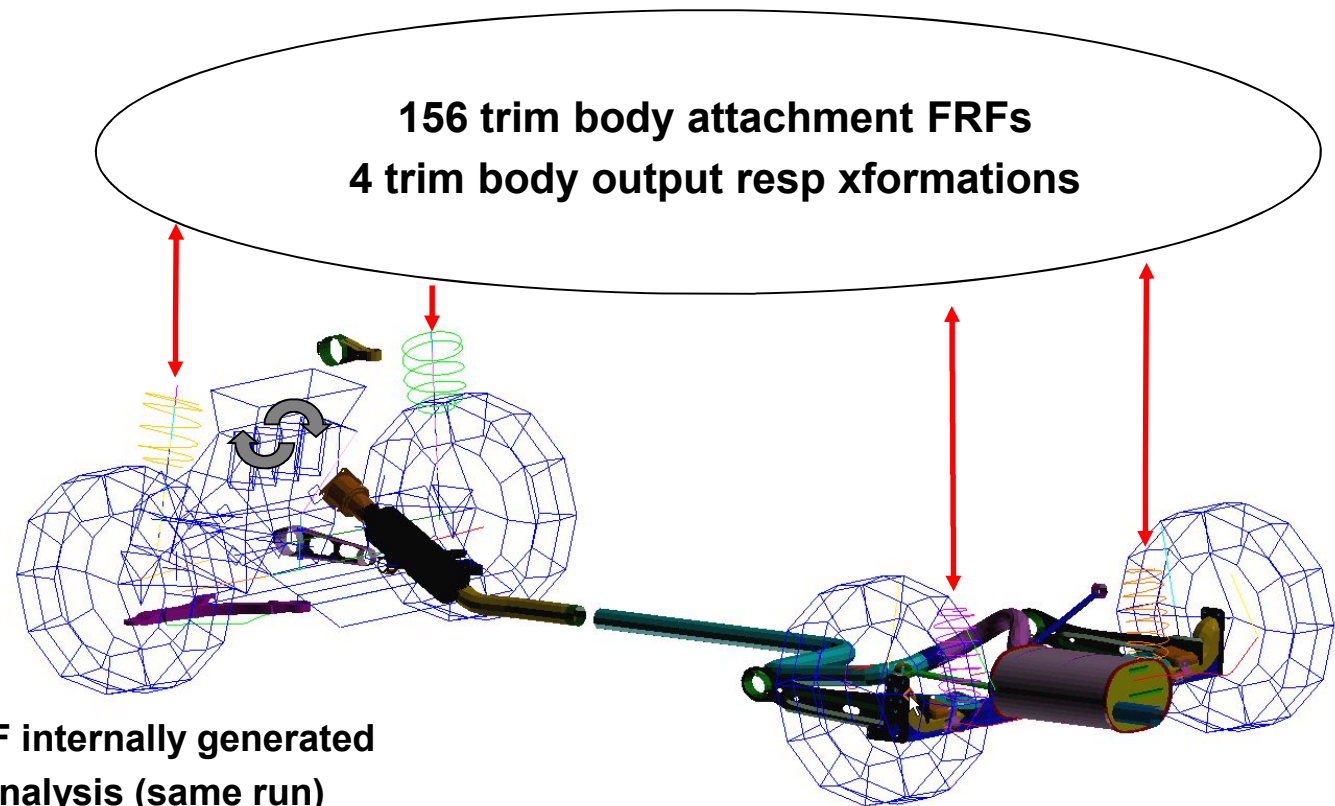
26 attachment points – 6 DOFs each – 156 FRFs

281 excitation frequencies

1.4 hrs smp=2 (one-time)

FRF/FBA Example 2 (cont.)

FRF Assembly Analysis – Suspension System + Trim Body FRF



Suspension system FRF internally generated

TB + Suspension FBA analysis (same run)

Equal & Opposite Engine Torque (user loads)

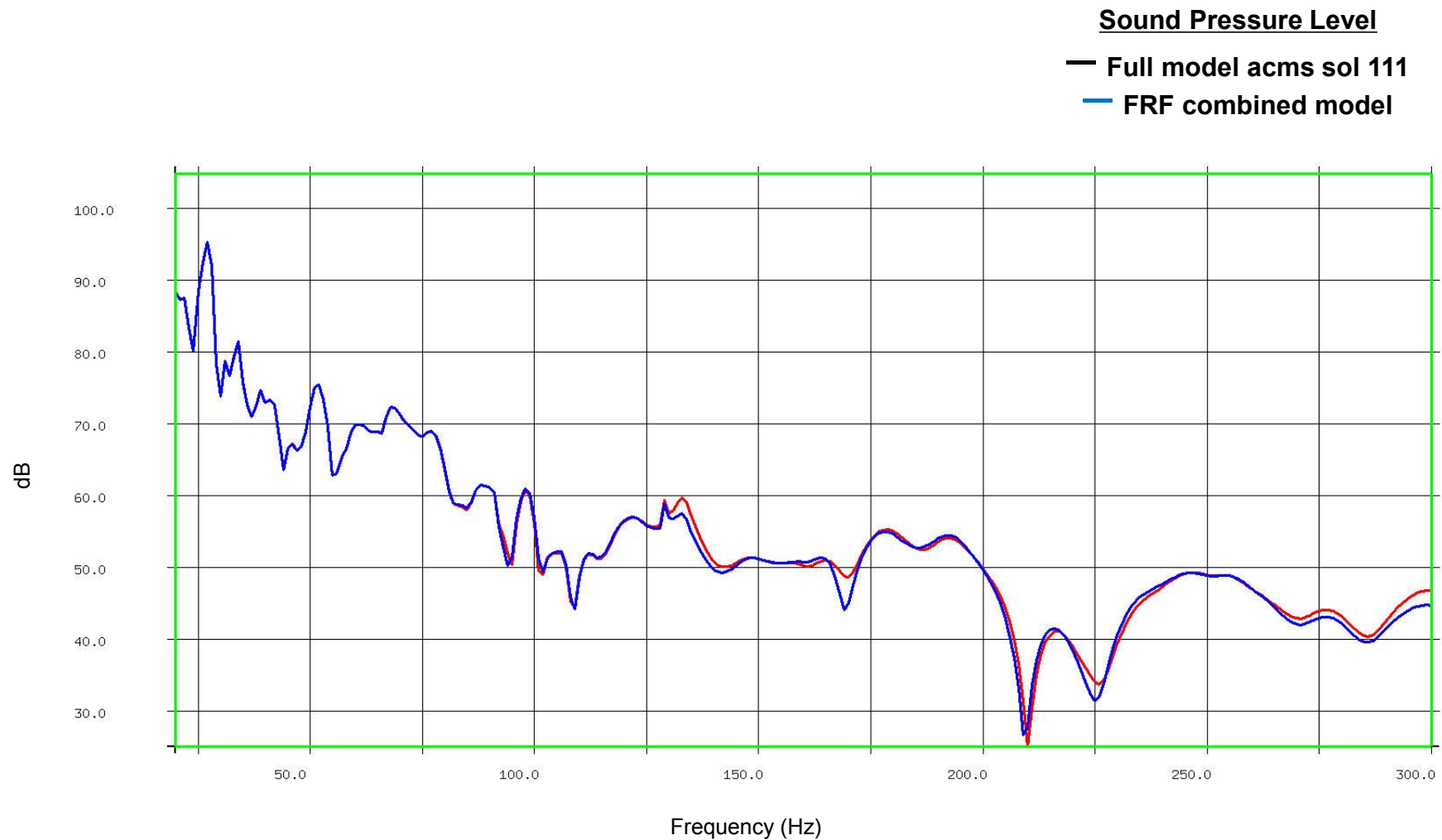
12 suspension / tb boundary responses

4 spindle suspension interior responses

4 interior TB full user load responses recovered

FRF/FBA Example 2 (cont.)

Driver Ear Acoustic Response

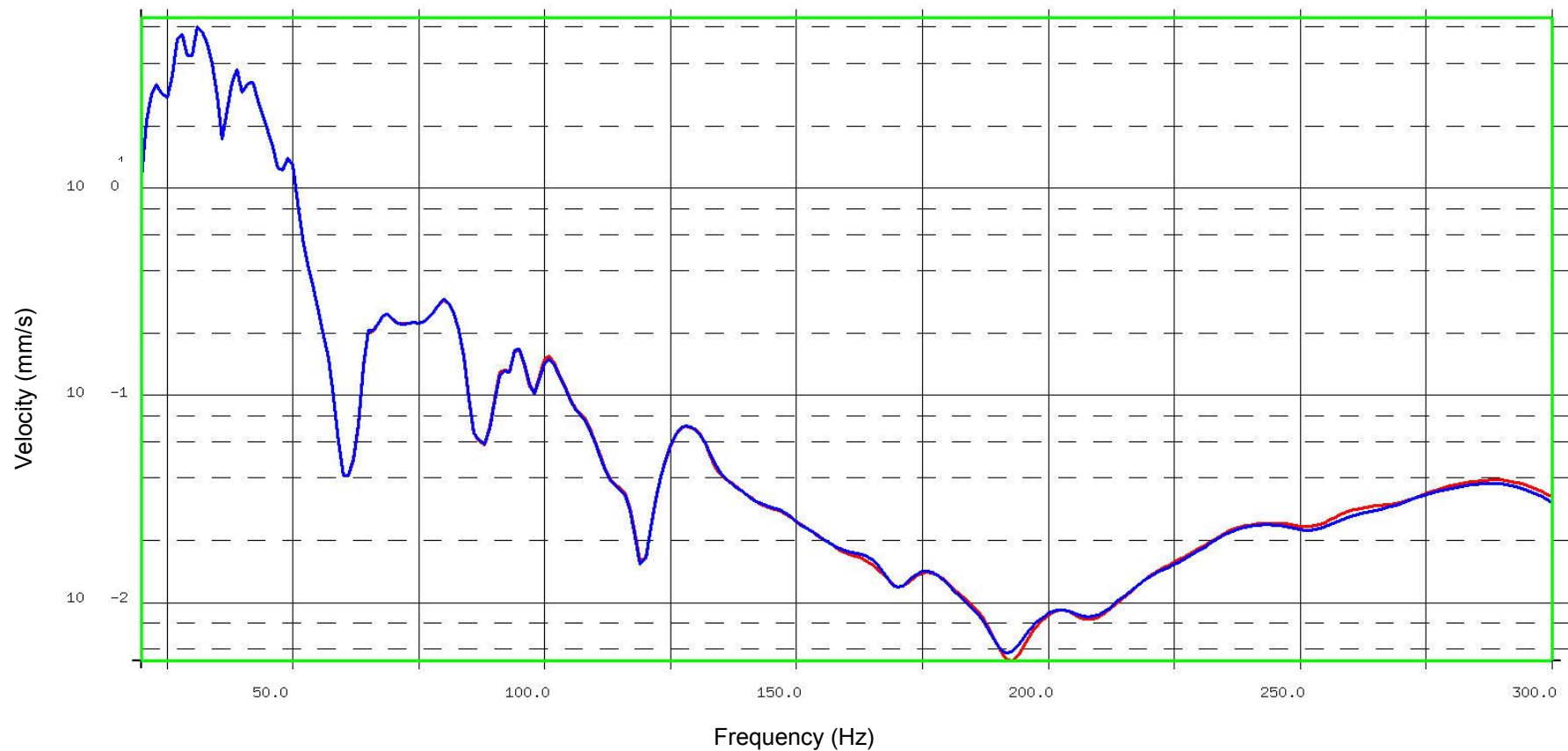


FRF/FBA Example 2 (cont.)

Steering Wheel Vibration

Resultant Velocity

— Full model acms sol 111
— FRF combined model



Example 3 – Include FRF from Testing

- **This example is the double-headed flyswatter with 8 components**
- **7 of the components are internal FRFs generated the same way as before**
- **1 component from test**
 - This is made-up component, not a real test component

Example 3 – Include FRF from Testing (cont.)

- **Typical FRF generation run for one of the components**

```
ASSIGN OUTPUT2='frfswt22_op2' UNIT=26 DELETE
ID FRFSWT22 - FRF TEST FOR FLYSWATTER MODEL USING THE OP2 OPTION
$
SOL      108
DIAG 8,15
CEND
TITLE = FRFSWT22 - FRF TEST FOR FLYSWATTER MODEL USING THE OP2 OPTION
SUBTITLE = FRF GENERATION FOR COMPONENT NO. 2
SET 1000 = 41,42
SET 2000 = 41,42,104
FRF (COMPID=2 COMPNAME=RIGHTTOP CONNPTS=1000 OP2=26)
FREQ = 5000
DISP = 2000
OLOAD=ALL
BEGIN BULK
PARAM,G,0.01
FREQ,5000,10.0,20.0,30.0
FRFXIT1,3,104
$
INCLUDE 'prop.blk'
INCLUDE 'se2.blk'
$
ENDDATA
```

Example 3 – Include FRF from Testing (cont.)

- **Assembly Run**

```
$
$   FRF COMPONENT FROM TESTING
$
$   ASSIGN UNVFILE='frfcomp1.unv' UNIT=25
$
$   7 FRF COMPONENTS FROM ANALYSIS
$
$   ASSIGN INPUTT2='frfswt22_op2' UNIT=26
$   ASSIGN INPUTT2='frfswt23_op2' UNIT=27
$   ASSIGN INPUTT2='frfswt24_op2' UNIT=28
$   ASSIGN INPUTT2='frfswt25_op2' UNIT=29
$   ASSIGN INPUTT2='frfswt26_op2' UNIT=30
$   ASSIGN INPUTT2='frfswt27_op2' UNIT=31
$   ASSIGN INPUTT2='frfswt28_op2' UNIT=32
$
$   SOL      108
$   DIAG 8,15
$   CEND
$   TITLE = FRFSWT2T - FRF BASED ASSEMBLY (FBA)
$   SUBTITLE = TEST FRF COMPONENT 1 WITH OTHER NASTRAN FRF COMPONENTS
$   FRF(ASM ASMOUT=ALL LOADLBL=ALT)
$   DISP = ALL
$   STRE = ALL
$   BEGIN BULK
$   FRFCOMP,1,LEFTTOP,UF,25,1000.0
$   INCLUDE 'frfswt22.asm'
$   INCLUDE 'frfswt23.asm'
$   INCLUDE 'frfswt24.asm'
$   INCLUDE 'frfswt25.asm'
$   INCLUDE 'frfswt26.asm'
$   INCLUDE 'frfswt27.asm'
$   INCLUDE 'frfswt28.asm'
$   ENDDATA
```

From test results

From MSC Nastran
runs

Example 4 – User Applied Load

```
assign inputt2='frf_18a_op2',unit=31
assign inputt2='frf_18b_op2',unit=32
assign inputt2='frf_18c_op2',unit=33
$
SOL 108
DIAG 8,15
CEND
subcase 1100
TITLE = sol 108 - bring in comp 10, 20, 30 using actual applied loads
SUBTITLE = 4 subcase output - 3 individual loads + combined loads
dload = 1100
FRF(ASM XITOUT=usertotl)
DISP(phase) = ALL
STRE(phase) = ALL
BEGIN BULK
dload,1100,1.,1.,101,1.,102,1.,104
$
rload2,101,101,,100
fbalload,101,comp10,93,3,1.
$
rload2,102,102,202,302,100
fbalload,102,20,104,3,1.
$
rload2,104,104,204,304,100
fbalload,104,comp30,8,3,1.
$
tabled1,100
,0.,1.,100.,1.,endt
$
include 'frf_18a.asm'
include 'frf_18b.asm'
include 'frf_18c.asm'
ENDDATA
```


WORKSHOP 5

- Perform Workshop 5: FRF

Interface Component Load (ICF)

- Starting in MSC Nastran 2013+, you can also calculate the interface component load (ICF)
- The ICFs are helpful in understanding the load path in an assembly
- Useful for joint design
- This is activated by a combination of the ICFGEN, ICFUSE, and ICFAUTO Case Control Command
- To obtain the ICF output, use the ICF Case Control Command

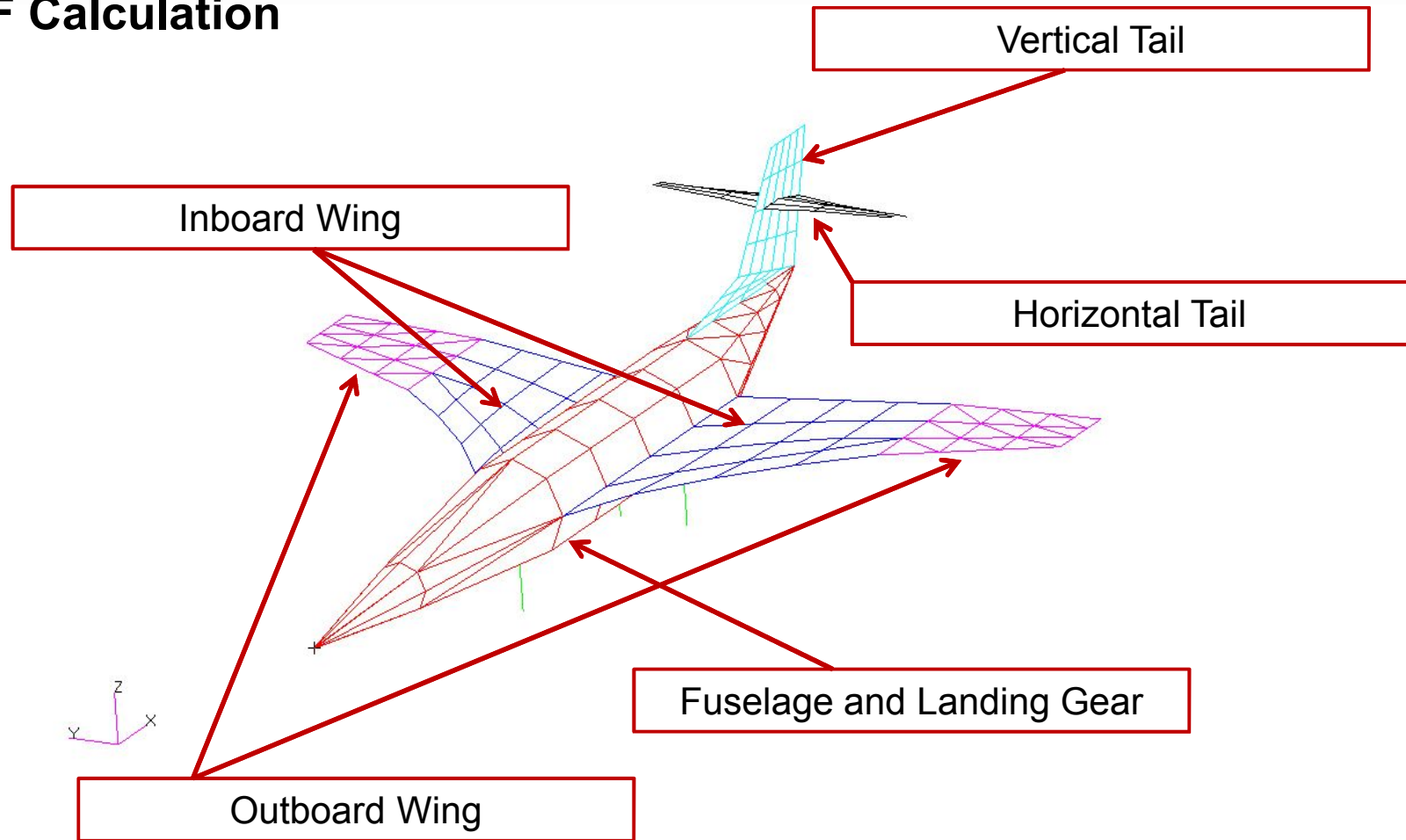
$$\text{ICF} \left(\begin{bmatrix} \text{PHYSICAL} \\ \text{MODAL} \end{bmatrix}, \begin{bmatrix} \text{PRINT, PUNCH} \\ \text{PLOT} \end{bmatrix}, \begin{bmatrix} \text{REAL or IMAG} \\ \text{PHASE} \end{bmatrix} \right) = \left\{ \begin{array}{c} \text{ALL} \\ " \\ \text{NONE} \end{array} \right\}$$

Examples:

```
ICF = 5
ICF (SORT2, PRINT, PUNCH, IMAG) = ALL
ICF (SORT2) = 20
ICF = ALL
```

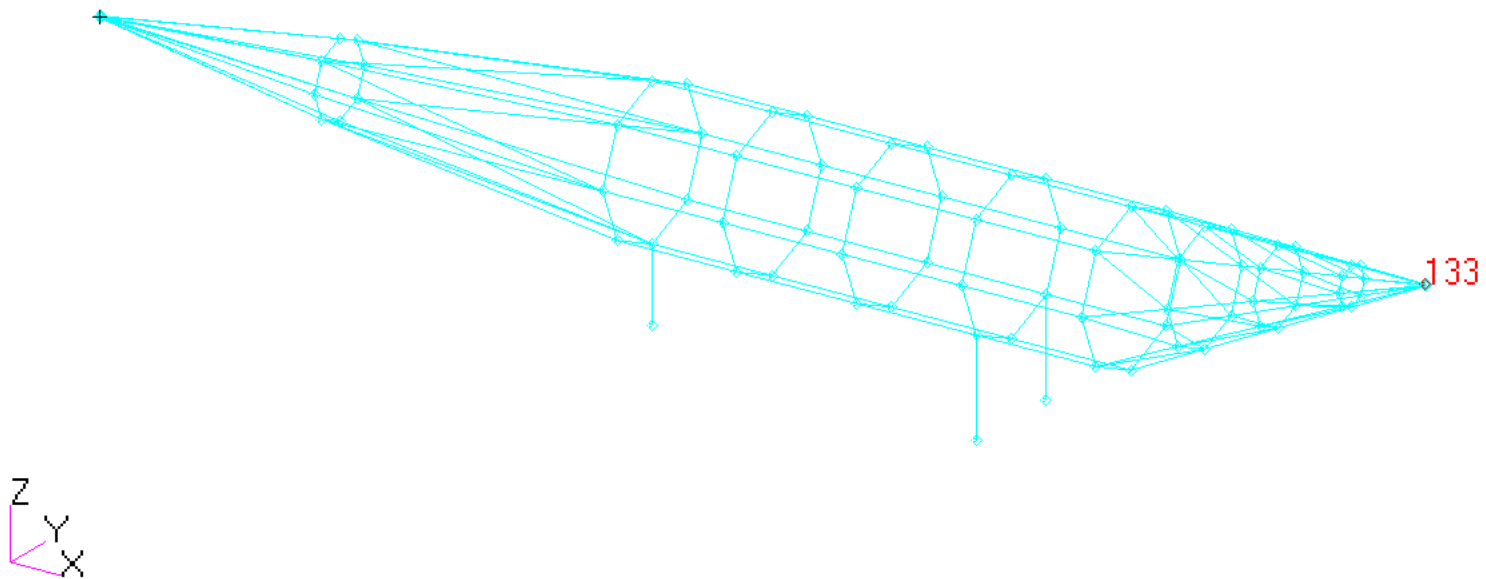
EXAMPLE 5

- ICF Calculation



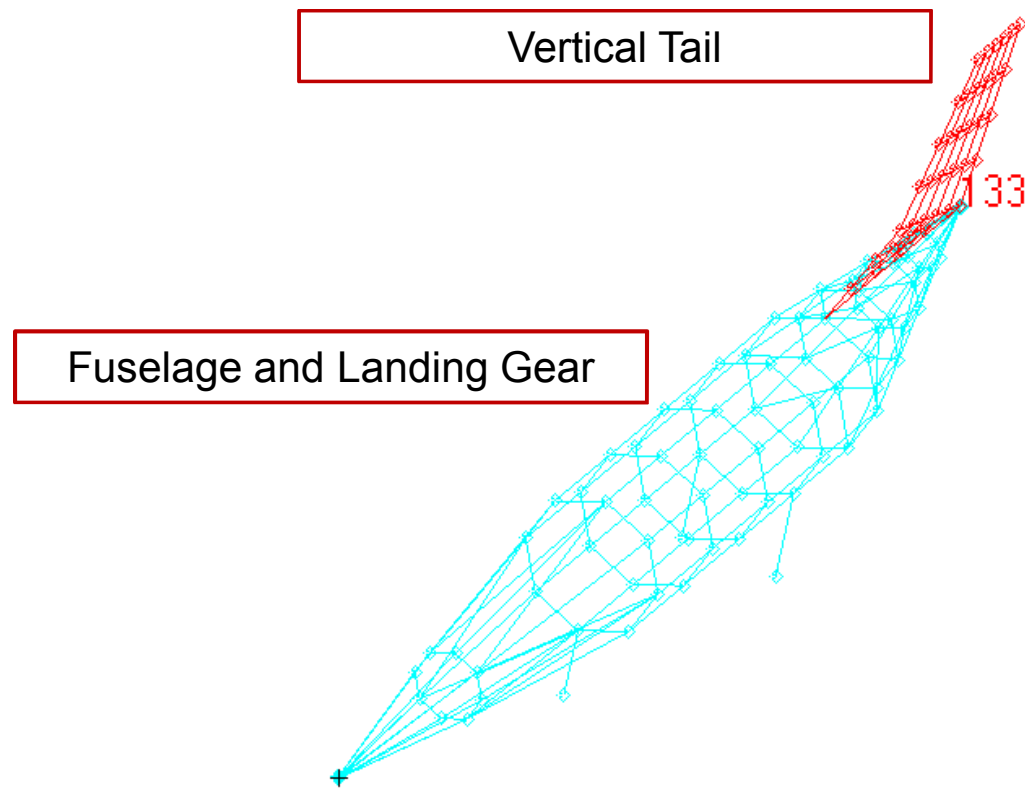
EXAMPLE 5 (cont.)

- Fuselage and Landing Gear



EXAMPLE 5 (cont.)

- Calculate ICF at Grid Point 133



EXAMPLE 5 (cont.)

- **FRF Generation run for fuselage+landing gear**
 - Similar setup for the other four components

```
ASSIGN OUTPUT2='frfop2-1.op2' UNIT=23 DELETE
$ FRF_FUSELAGE - FRF FBA For Airplane
$
SOL 108
CEND
ECHO = NONE
$
SUBCASE 1
TITLE = FRF_FUSELAGE - FRF GEN
SUBTITLE = FRF Component 1 - FUSELAGE
$
SET 200= 55,THRU,59,35,THRU,39,24,106,115,124,133
SET 300= 9,14,19,55,THRU,59,35,THRU,39,24,106,115,124,133,89
SPC = 2
$
FRF(COMPID=1 COMPNAME=FUSELAGE CONNPTS=200 op2=23)
$
DISPLACEMENT=300
$
FREQUENCY = 1
$
```

```
BEGIN BULK
$! Bulk Data Pre Section
PARAM POST 1
$! Bulk Data Model Section
$PARAM PRTMAXIM YES
$
$ Loads for Load Case : Untitled.SC1
SPCADD 2 1
$ Displacement Constraints of Load Set : spc1.1
SPC1 1 123456 457 458 459
$
$
FREQ1 1 10. 10. 3
$
$ Component1: Fuselage and landing gear
$
include 'fuse_lndgr-1.bdf'
$
ENDDATA
```

EXAMPLE 5 (cont.)

- **ICF for fuselage (component 1) and vertical tail (component 3)**
 - Using OP2 Option and request ICF at grid 133

```
$
ASSIGN INPUTT2='frfop2-1.op2' UNIT=23
ASSIGN INPUTT2='frfop2-2.op2' UNIT=24
ASSIGN INPUTT2='frfop2-3.op2' UNIT=25
ASSIGN INPUTT2='frfop2-4.op2' UNIT=26
ASSIGN INPUTT2='frfop2-5.op2' UNIT=27
$
ASSIGN OUTPUT2='frf-icf.op2',unit=35
$
ID FRF_ASSEMBLY - USING OP2
$
SOL 108
DIAG 8,15
CEND
ECHO=NONE
TITLE = FBA USING THE ASM OPTION
$
SET 2 = 133
SET 100 = 1,3
DLOAD=20
FRF(ASM ICFAUTO=100 icfop2=35)
ICF=2
$
```

```
$
BEGIN BULK
$
RLOAD1,20,100,,,2
$
FBALOAD,100,IB_WINGS,147,3,-95.0
FBALOAD,100,IB_WINGS,352,3,-28.0
FBALOAD,100,HOR_TAIL,330,3,-72.0
$
TABLED1,2
,0.0,1.0,100.0,1.0,ENDT
PARAM,POST,-1
$
include 'frf-gen-op2-1.asm'
include 'frf-gen-op2-2.asm'
include 'frf-gen-op2-3.asm'
include 'frf-gen-op2-4.asm'
include 'frf-gen-op2-5.asm'
$
ENDDATA
```

EXAMPLE 5 (cont.)

- Interface Component load from the fuselage side

```

^^^ICF COMPUTATION FOR FRF COMPONENT 1 (FUSELAGE)
^^^
1 FBA USING THE ASM OPTION
JUNE 18, 2013 MSC Nastran 5/30/13 PAGE 9
FBA OUTPUT FOR FRF COMPONENT 1 (FUSELAGE)
0 TOTAL USER LOAD (USER SUBCASE/DLOAD= 1/ 20) FBA SUBCASE 19999
FREQUENCY = 1.000000E+01
COMPLEX INTER-COMPONENT FORCE VECTOR
(REAL/IMAGINARY)
POINT ID. TYPE T1 T2 T3 R1 R2 R3
0 133 G -1.795020E+01 2.177065E+01 -3.258662E+01 2.849422E+01 -8.328401E-04 -1.607762E+00
1 FBA USING THE ASM OPTION
JUNE 18, 2013 MSC Nastran 5/30/13 PAGE 10
FBA OUTPUT FOR FRF COMPONENT 1 (FUSELAGE)
0 TOTAL USER LOAD (USER SUBCASE/DLOAD= 1/ 20) FBA SUBCASE 19999
FREQUENCY = 2.000000E+01
COMPLEX INTER-COMPONENT FORCE VECTOR
(REAL/IMAGINARY)
POINT ID. TYPE T1 T2 T3 R1 R2 R3
0 133 G -1.966426E+01 9.091413E+00 -3.628968E+01 4.409430E+01 -9.194112E-04 -2.653217E+00
1 FBA USING THE ASM OPTION
JUNE 18, 2013 MSC Nastran 5/30/13 PAGE 11
FBA OUTPUT FOR FRF COMPONENT 1 (FUSELAGE)
0 TOTAL USER LOAD (USER SUBCASE/DLOAD= 1/ 20) FBA SUBCASE 19999
FREQUENCY = 3.000000E+01
COMPLEX INTER-COMPONENT FORCE VECTOR
(REAL/IMAGINARY)
POINT ID. TYPE T1 T2 T3 R1 R2 R3
0 133 G -2.386446E+01 -2.258092E+02 -4.488052E+01 2.461727E+02 -1.120235E-03 -1.666065E+01
1 FBA USING THE ASM OPTION
JUNE 18, 2013 MSC Nastran 5/30/13 PAGE 12
FBA OUTPUT FOR FRF COMPONENT 1 (FUSELAGE)
0 TOTAL USER LOAD (USER SUBCASE/DLOAD= 1/ 20) FBA SUBCASE 19999
FREQUENCY = 4.000000E+01
COMPLEX INTER-COMPONENT FORCE VECTOR
(REAL/IMAGINARY)
POINT ID. TYPE T1 T2 T3 R1 R2 R3
0 133 G -3.485371E+01 1.873639E+02 -6.801958E+01 -4.977777E+01 -1.649792E-03 4.298093E+00

```


EXAMPLE 5 (cont.)

- Interface Component load from the vertical tail side

```

1  (TOTAL DATA WRITTEN FOR DATA BLOCK = 4760 WORDS.)
0  FBA USING THE ASM OPTION
0  TOTAL USER LOAD
    FREQUENCY = 1.000000E+01
    COMPLEX INTER-COMPONENT FORCE VECTOR
    (REAL/IMAGINARY)
    POINT ID. TYPE T1 T2 T3 R1 R2 R3
    133 G 1.795020E+01 -2.177065E+01 3.258662E+01 -2.849422E+01 8.328401E-04 1.607762E+00
    0.0 0.0 0.0 0.0 0.0 0.0
1  FBA USING THE ASM OPTION
0  TOTAL USER LOAD
    FREQUENCY = 2.000000E+01
    COMPLEX INTER-COMPONENT FORCE VECTOR
    (REAL/IMAGINARY)
    POINT ID. TYPE T1 T2 T3 R1 R2 R3
    133 G 1.966426E+01 -9.091413E+00 3.628968E+01 -4.409430E+01 9.194112E-04 2.653217E+00
    0.0 0.0 0.0 0.0 0.0 0.0
1  FBA USING THE ASM OPTION
0  TOTAL USER LOAD
    FREQUENCY = 3.000000E+01
    COMPLEX INTER-COMPONENT FORCE VECTOR
    (REAL/IMAGINARY)
    POINT ID. TYPE T1 T2 T3 R1 R2 R3
    133 G 2.386446E+01 2.258092E+02 4.488052E+01 -2.461727E+02 1.120235E-03 1.666065E+01
    0.0 0.0 0.0 0.0 0.0 0.0
1  FBA USING THE ASM OPTION
0  TOTAL USER LOAD
    FREQUENCY = 4.000000E+01
    COMPLEX INTER-COMPONENT FORCE VECTOR
    (REAL/IMAGINARY)
    POINT ID. TYPE T1 T2 T3 R1 R2 R3
    133 G 3.485371E+01 -1.873639E+02 6.801958E+01 4.977777E+01 1.649792E-03 -4.298093E+00
    0.0 0.0 0.0 0.0 0.0 0.0

```


SECTION 10

Nonlinear Transient Analysis

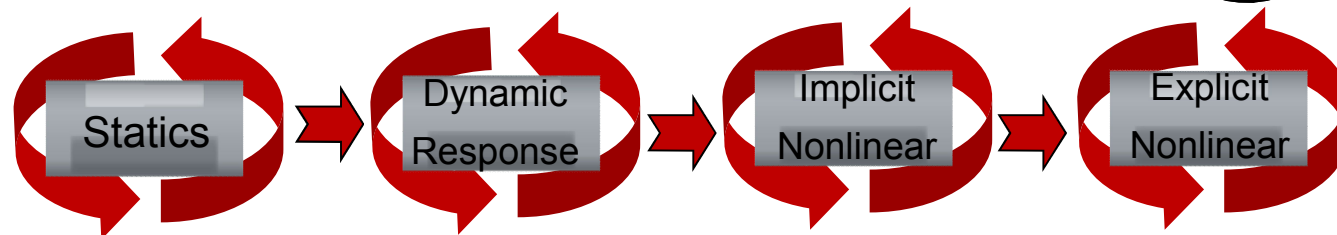
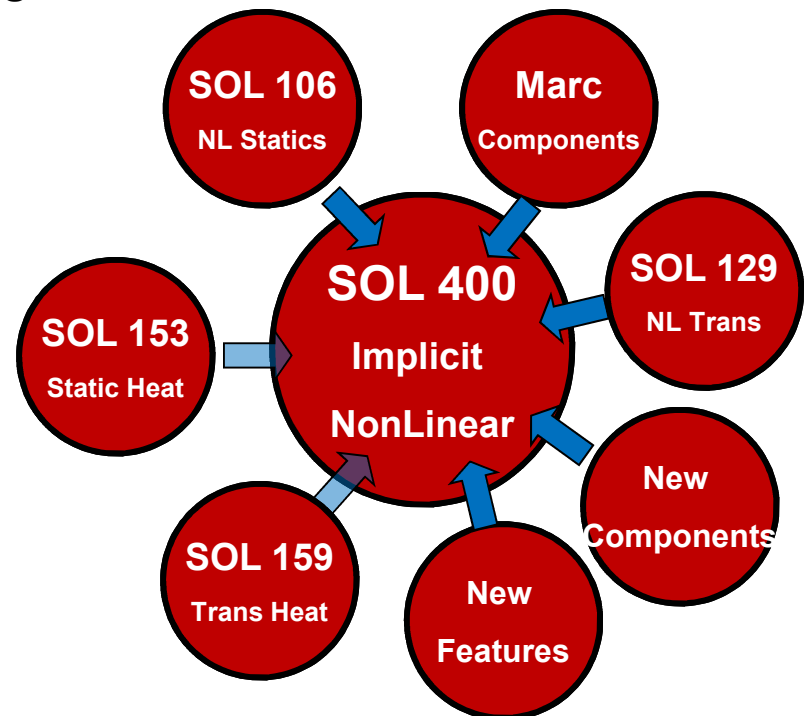
Objectives

- **Introduction to SOL400 (Nastran Nonlinear)**
- **Review of Nonlinear versus Linear**
- **Review of Nonlinear Solution Strategies**
- **Introduction to Nonlinear Transient Dynamics**

What is SOL400 ?

- **New advanced nonlinear solution process**

- Combines capabilities of multiple solution sequences and software components into a common solution
- Allows for analysis Chaining
 - Automatically chaining together sequences of analyses with output state of one used as input state for another
 - Model complete processes in a single simulation through analysis chaining



Advanced Nonlinear

- **Advanced Integrated Nonlinear**
 - **Utilize "native" Nastran elements**
 - No translation required
 - **Combine static & transient in one analysis**
 - Pre-stress, transient, steady-state analysis chaining
 - Thermal-Structural analysis chaining
 - Multiple, independent loadcases in 1 run
 - Linear Perturbation
 - **Use general contact capability**
 - Solid-to-solid, surface-to-surface, edge-to-edge, beam-to-beam, etc.
 - **Go beyond small-strain element limitations**
 - Large strain elements / materials (shells, solids)

Advanced Nonlinear (cont.)

- **Advanced Integrated Nonlinear**
 - **Model large displacement / rotation rigid elements**
 - Kinematic RBEi elements
 - **Nonlinear Connectors**
 - Large displacement / rotation CFAST, CBUSH, and CWELD elements
 - “Fuse” capability of CBUSH element
 - **Simulate Composite Progressive Failure**
 - Virtual Crack Closure Technique (VCCT)
 - Cohesive Zone Modeling
 - Progressive Failure Analysis (PFA)
 - Genoa Micromechanical modeling
 - Composite Beam

Nonlinear versus Linear

FEM Quantities in Linear Analysis

- Kinematics**

$$\mathbf{u}_e = \mathbf{T}_{eg} \cdot \mathbf{u}_g$$

**Element
Displacement
Vector** **Displacement
Transformation
Matrix** **Global
Displacement
Vector**

- Compatibility**

$$\boldsymbol{\varepsilon} = \mathbf{B} \cdot \mathbf{u}_e$$

**Element
Strains** **Strain
Displacement
Matrix** **Element
Displacement
Vector**

- Constitutive Law**

$$\boldsymbol{\sigma} = \mathbf{D} \cdot \boldsymbol{\varepsilon}$$

**Element
Stresses** **Stress-Strain
Relationship** **Element
Strains**

FEM Quantities in Linear Analysis (Cont.)

- **Equilibrium**

$$\mathbf{P} = \sum \mathbf{T}_{eg}^T \cdot \mathbf{F}_e$$

External Force Element
Load Vector Transformation Forces
Matrix

- **Constraints**

$$\mathbf{u}_g = \alpha$$

- **The transformation matrices do not change**
- **Force equals displacement transformation**
- **The constraints (SPC, MPC) do not change**

Linear Analysis Consequences

- **Solving a Linear System of Equations**

$$\mathbf{K} \cdot \mathbf{u} = \mathbf{P}$$

- **In Linear Analysis it follows that:**
 - Loads are independent of deformation
 - Displacements are directly proportional to the loads
 - Results for different loads can be superimposed

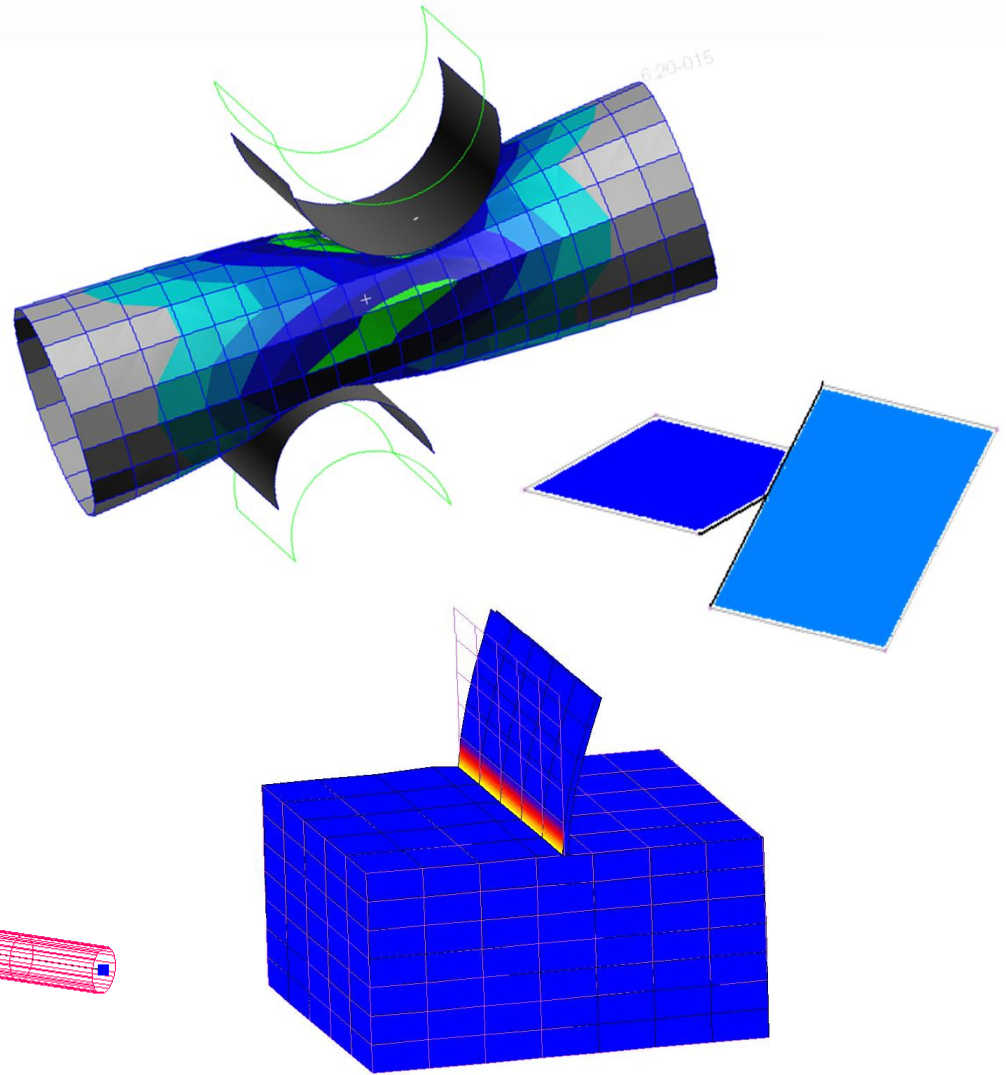
Nonlinear Analysis

- **In Nonlinear Analysis, upon deformation the following changes are allowed:**
 - $T_{(disp)}$ geometric nonlinear
 - $T_{(force)}$ follower forces
 - B large strain
 - D material nonlinear
 - $T_{(disp)} \neq T_{(force)}$ buckling
 - α constraints, contact
- **It follows that the system of equations gets nonlinear, the load sequence is unique and the results must not be superimposed**

Sources of Nonlinearities

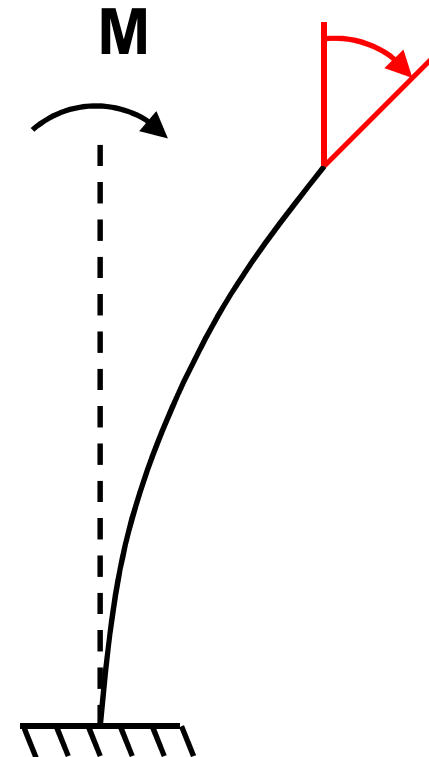
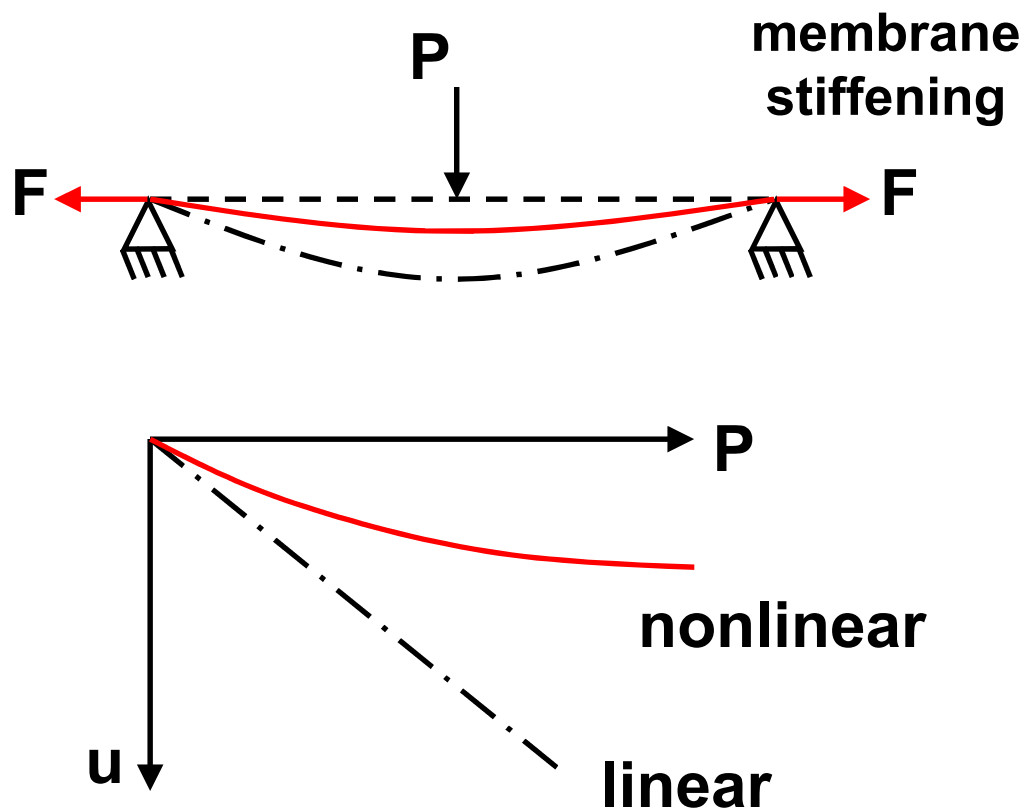
Contact and Constraint Changes

- **Surface to Surface**
- **Edge to Surface**
- **Edge to Edge**
- **Beam to Beam**
- **Rigid to Deformable**
- **SPC / MPC**



Geometric Nonlinearity

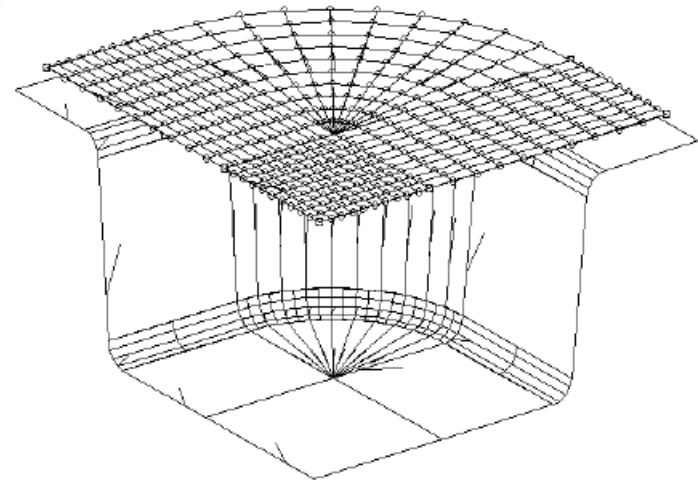
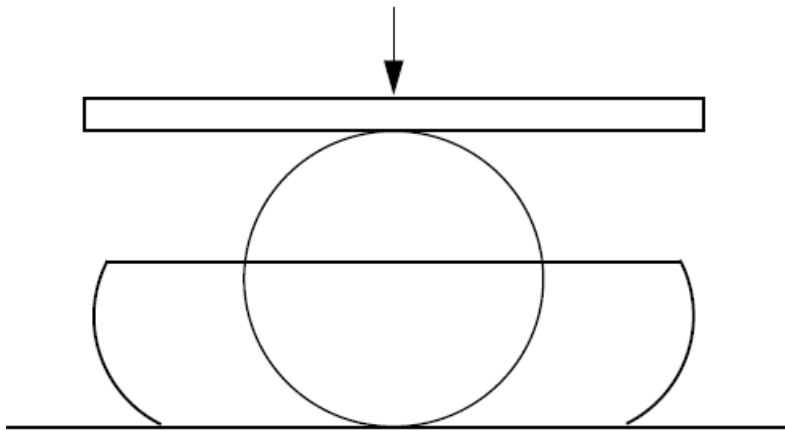
- Large displacements and rotations, but small strains



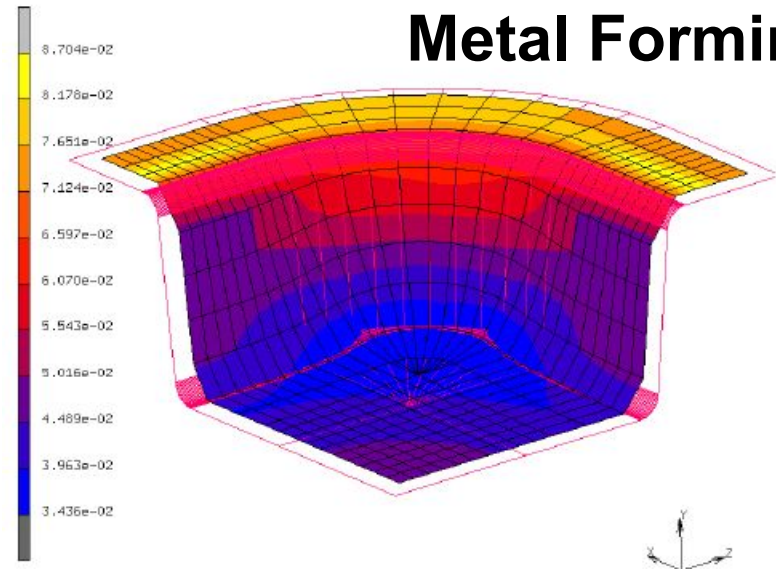
Geometric Nonlinearity (cont.)

- Large displacement and rotation and large strains

Rubber Bearing

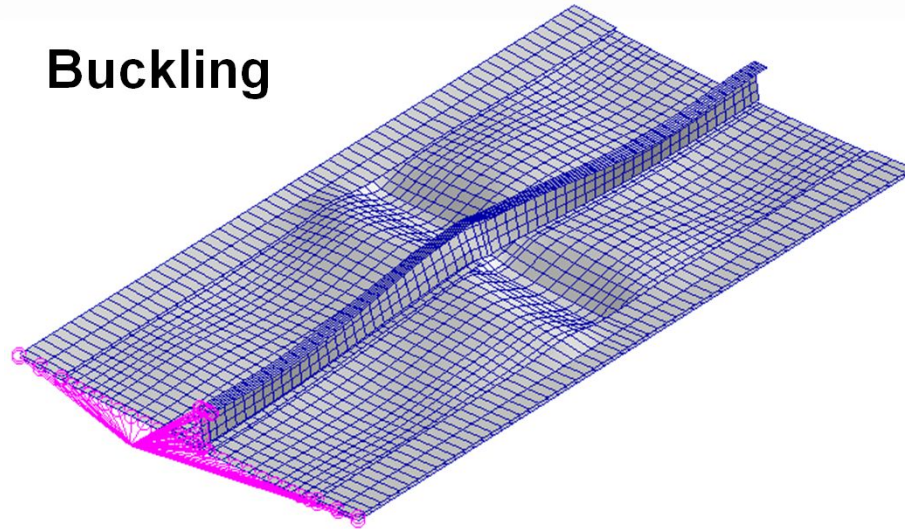


Metal Forming

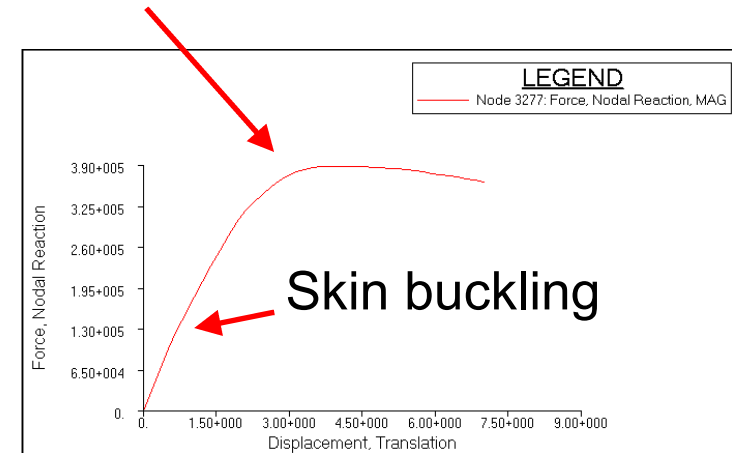


Geometric Nonlinearity (cont.)

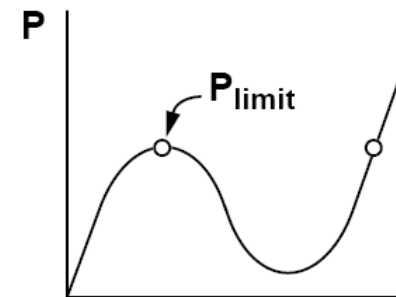
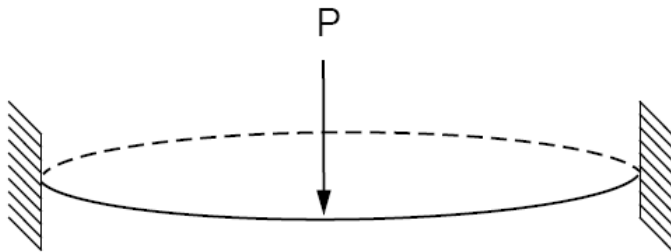
- **Buckling**



Panel failure

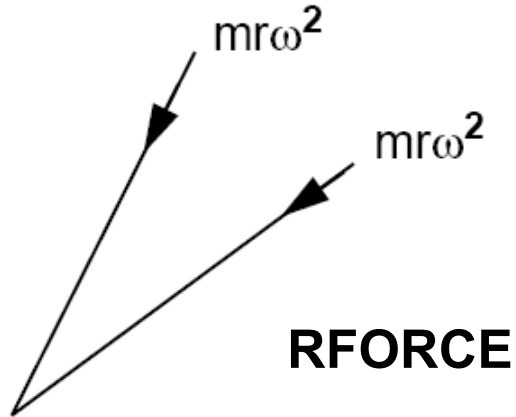


- **Snap-Through**

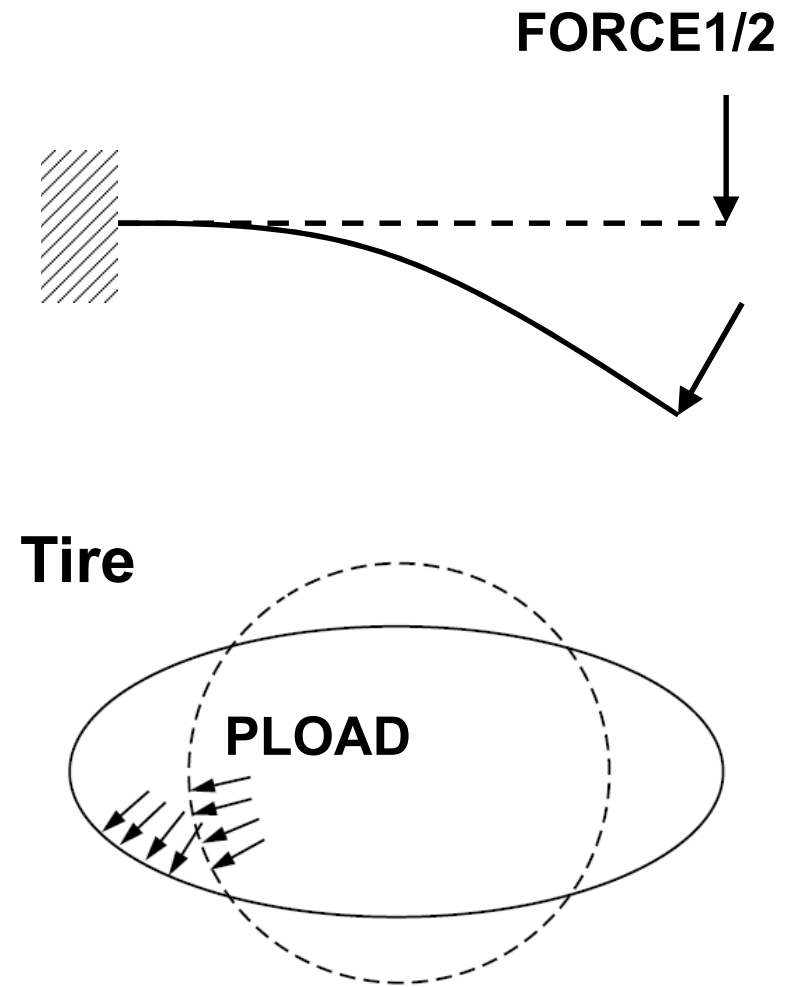


Geometric Nonlinearity (cont.)

- **Follower Forces**



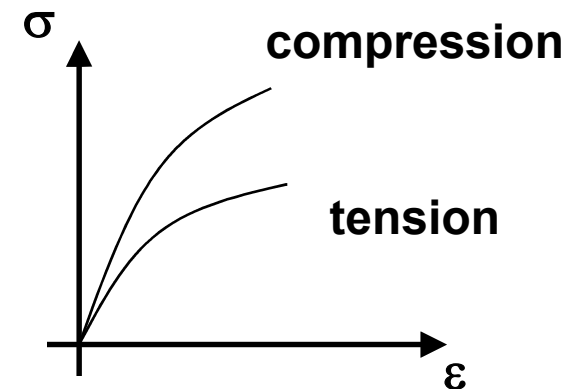
Temperature Loads



Material Nonlinearity

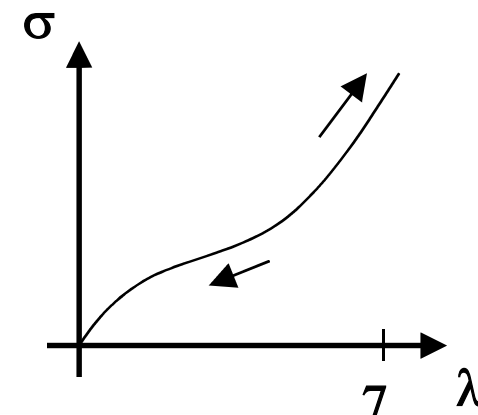
- **Nonlinear Elastic**

- small strains
- different curves for tension and compression
- after unloading structure is undeformed



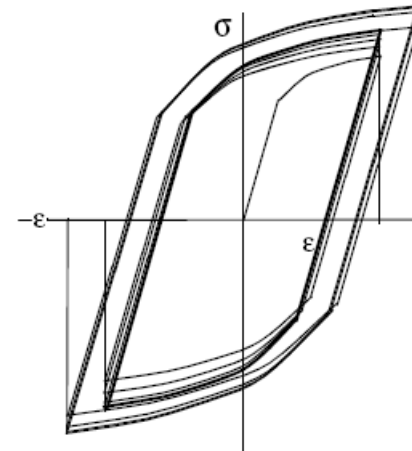
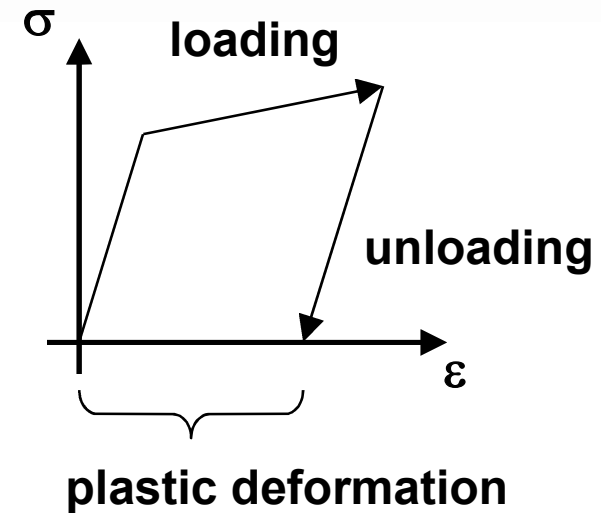
- **Hyperelastic**

- normally large strain
- Poisson's ratio close to 0.5
- mainly rubber



Material Nonlinearity (cont.)

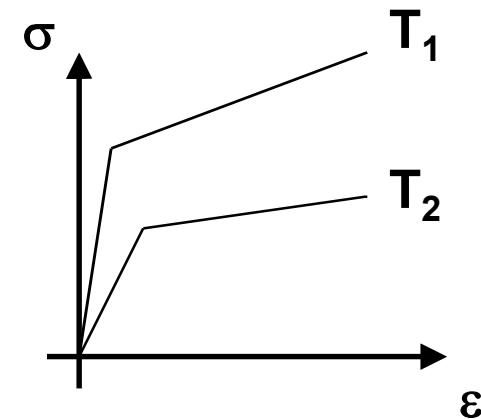
- **Elastic-Plastic**
 - small and large strain
 - isotropic, anisotropic, pressure dependent
 - initial stress and plastic strain
- **Cyclic Plasticity**
 - yield stress changes with the number of cycles
 - based on the work of Chaboche



Material Nonlinearity (cont.)

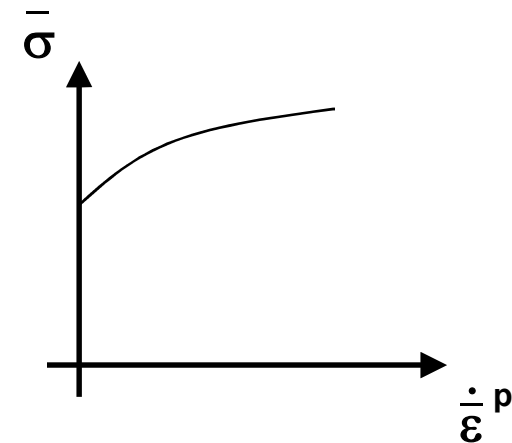
- **Temperature Dependent**

- stress–strain curve depends on temperature
- applies to each type of material nonlinearity



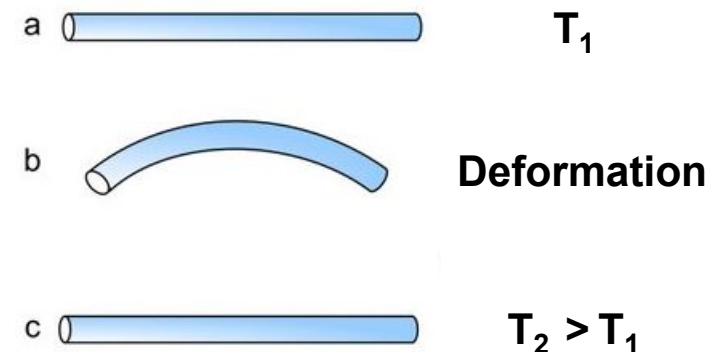
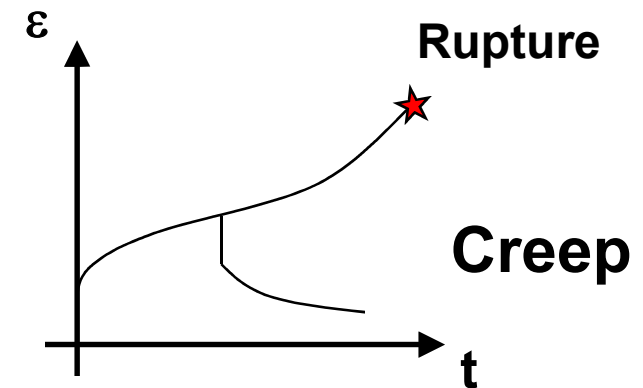
- **Rate Dependent**

- yield stress depends on equivalent plastic strain rate
- applies to elastic-plastic material



Material Nonlinearity (cont.)

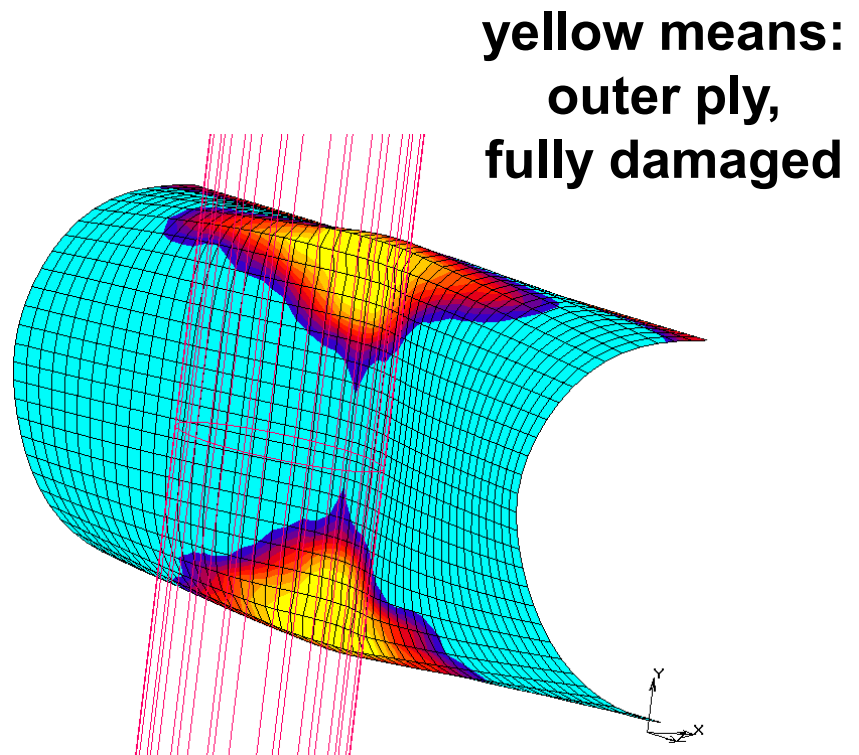
- **Time Dependent**
 - Material properties change with time
 - Creep and relaxation
 - Visco-Elasticity and Visco-Plasticity
- **Shape Memory**
 - Material properties depend on crystal structure (Martensite & Austenite)
 - Phase changes due to temperature and stresses



Material Nonlinearity (cont.)

- **Composite Failure**

- Maximum Stress & Strain
- Hill
- Hoffmann
- Tsai-Wu
- Puck
- Hashin
- Progressive Failure
 - Puck
 - Hashin
 - Genoa Integration

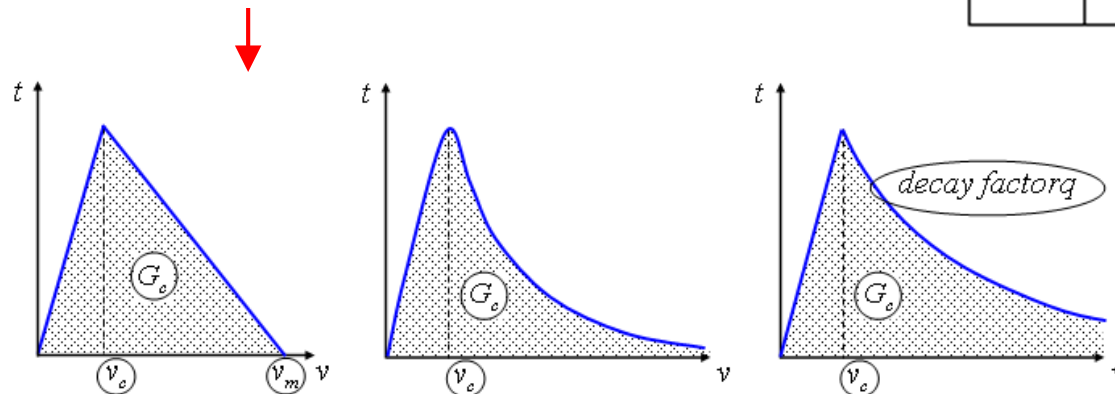
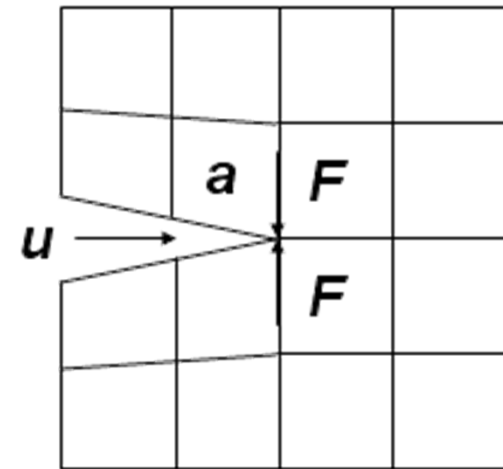


**Rigid elliptical cylinder hitting
composite shell**

Material Nonlinearity (cont.)

- **Fracture**

- Application: Delamination in Composite Analysis
- Virtual Crack Closure Technique (VCCT) →
- Cohesive Zone Modeling



SOL 400 Input File Example

```
ID LIEBE,TEST
SOL 400
DIAG 8
CEND
TITLE = THIS IS A DEMO INPUT EXAMPLE
SUBCASE 10
  STEP 1
    LOAD = 1
    NLPARM = 110
  STEP 2
    ANALYSIS = NLTRAN
    DLOAD = 3
    TSTEPNL = 130
BEGIN BULK
PARAM,  LGDISP, 1
$. . . . . 2 . . . . . 3 . . . . . 4 . . . . . 5 . . . . . 6 . . . . . 7 . . . . . 8 . . . . . 9 . . . . . 0
NLPARM  110      25              ITER    1          15          P          NO
+
MAT1    1        210000.         0.3      7.85-9    1.2-6
MATS1   1                PLASTIC 1000.    1          1          240.
.
```

SOL 400 Input File Example (cont.)

- **SOL 400 for Nonlinear Analysis**
 - Nonlinear Statics and Transient Analysis
 - Perturbation STEPs
- **Nonlinear Iteration Strategy**
 - NLPARM, Nonlinear Parameters for Statics
 - TSTEPNL, Nonlinear Parameters for Transient
 - NLSTEP, replaces NLPARM, TSTEPNL, NLPCI, NLADAPT
- **Geometric Nonlinear Analysis**
 - param, lgdisp, 1
- **Material Nonlinear Analysis**
 - MATS1, MATEP, for elastic-plastic Material

Documentation



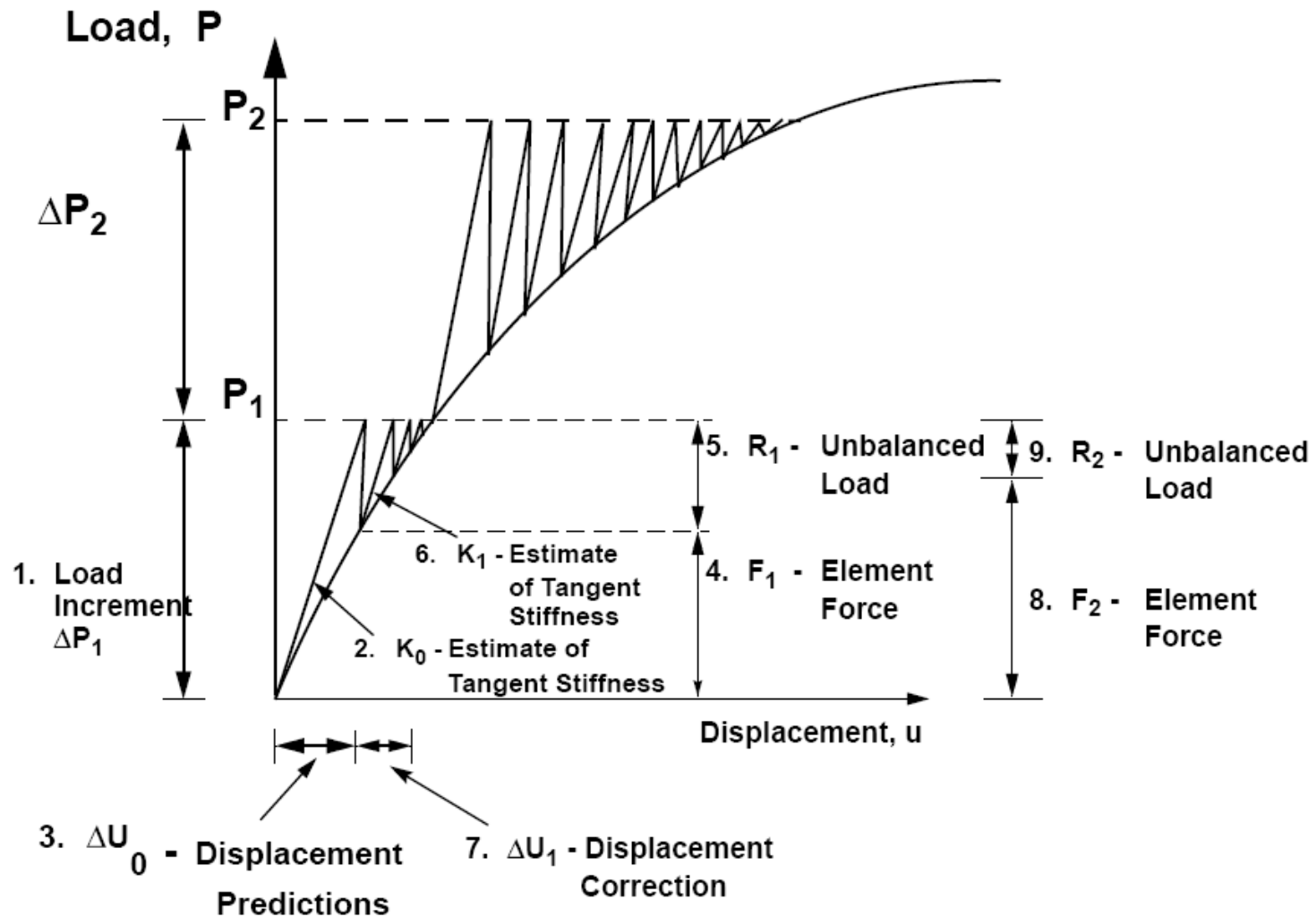
- **MSC Nastran**
 - MSC Demonstration Problems
 - contains examples for the efficient use of SOL 400
 - MSC Nastran Quick Reference Guide



- **Marc User's Manuals**
 - Volume A – Theory and User Information
 - Volume B – Element Library
 - Volume E – Demonstration Problems

Nonlinear Solution Strategy

Basic Concept of Nonlinear Iteration



Basic Concept of Nonlinear Iteration (cont.)

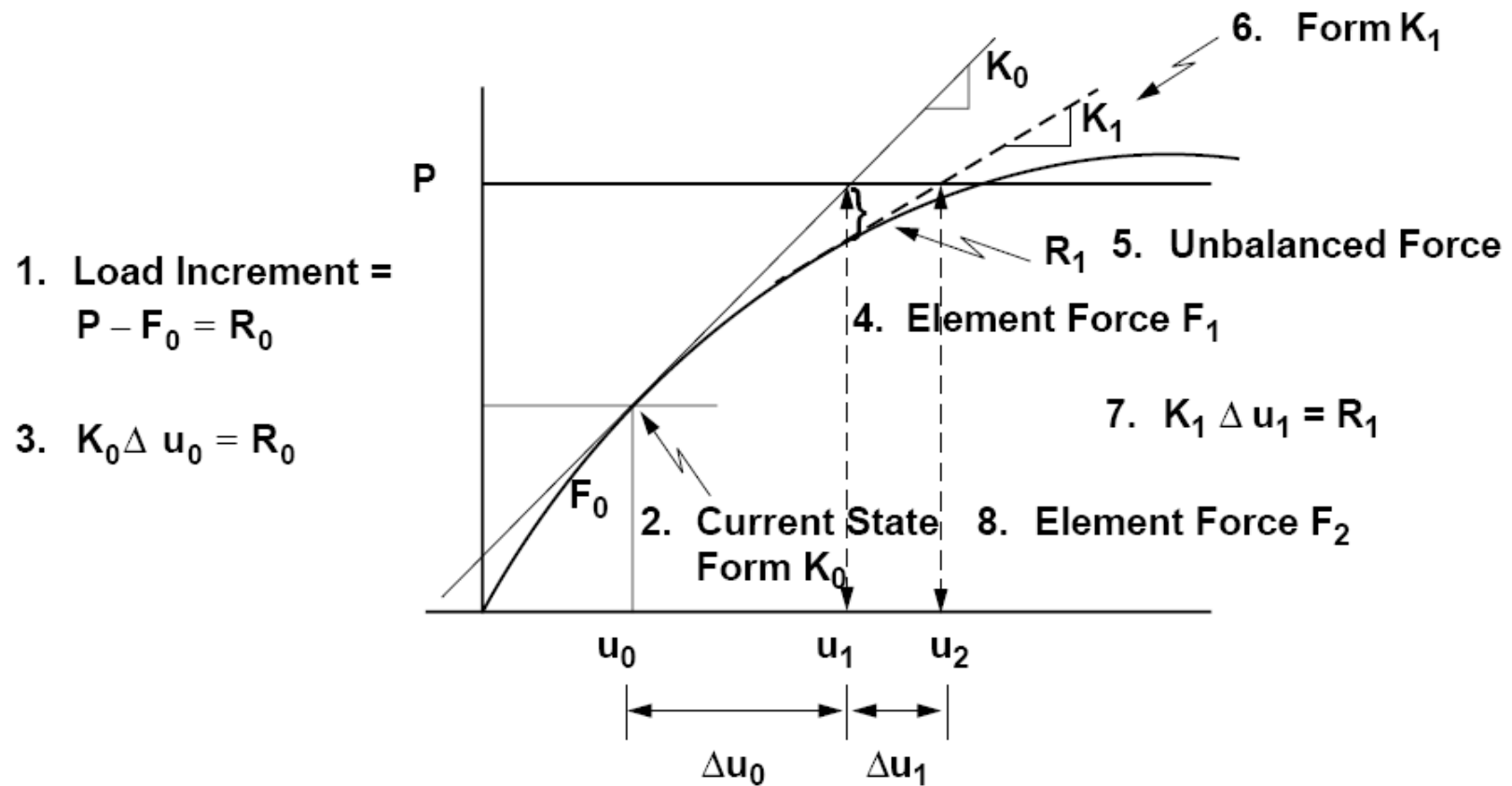
- Multistep Approach
 1. Determine an **increment** (e.g. load, displacement or arc length)
 2. Determine an estimate of the tangent stiffness matrix (default). The user can request that the stiffness matrix remains the same or will be updated exactly (tangent stiffness matrix).
 3. Determine the displacement increment
 4. Calculate the element forces
 5. Calculate the unbalanced load and check for convergence. If converged, go to Step 1

Basic Concept of Nonlinear Iteration (cont.)

- If not converged, continue as follows:
 6. Determine an estimate of tangent stiffness matrix
 7. Determine the displacement increment due to the unbalanced load
 8. Calculate the element resisting forces
 9. Calculate the unbalanced load and check for convergence. If converged, go to Step 1. If not converged, go to Step 6 or start the **divergence** procedure (explained later).

Full Newton Raphson Method

- Stiffness Update at **each** iteration



NLSTEP Bulk Data Entry

1	2	3	4	5	6	7	8	9	10
NLSTEP	ID	TOTTIME							
	"GENERAL"	MAXITER	MINITER	MAXBIS	CREEP				
	"FIXED"	NINC	NO						
	"ADAPT"	DTINITF	DTMINF	DTMAXF	NDESIR	SFACT	INTOUT	NSMAX	
		IDAMP	DAMP	CRITTID	IPHY'S	LIMTAR	RSMALL	RBIG	
		ADJUST	MSTEP	RB	UTOL				
	"ARCLN"	TYPE	DTINITFA	MINALR	MAXALR	SCALEA	NDESIR	NSMAXA	
	"HEAT"	CONVH	EPSUH	EPSPH	EPSWH	KMETHODH	KSTEPH		
		MAXQNH	MAXLSH	LSTOLH					
	"MECH"	CONV	EPSU	EPSP	EPSW	KMETHOD	KSTEP	MRCONV	
		MAXQN	MAXLS	LSTOL	FSTRESS				
	"COUP"	HGENPLAS	HGENFRIC						
	"RCHEAT"	SOLVER	DRLXCA	ARLXCA	BALENG	DAMPC	GRVCON	CSGFAC	
		NRLOOP	OUTINV	DTIME1					

- This entry contains all analysis disciplines

NLSTEP for Structural Analysis

- **The NLSTEP entry controls:**
 - Stiffness Update Strategy
 - Convergence Criteria
 - Convergence Tolerances
 - Frequency of Output
- **It can be used for**
 - Fixed Increment Stepping (FIXED)
 - Automatic Increment Stepping (ADAPT)
- **It can be used in Statics and Dynamics (in the past: NLPARM/TSTEPNL which are still available)**

Nonlinear Output Control

- **Example's .sts file:**

```
information summary of job: ./demo_400_nlio_std
version: MD Nastran 2011.1.0, Built on Apr 27, 2011
date: Jun 12, 2011; Day Time: 13:00:34
```

subcase /step #	inc #	cycl #	sepa #	cut #	cycl #	split #	separ #	cut #	rmesh #	time step of	total time of	wall time	cpu time	max resp.	type
1															
1	0	0	0	0	0	0	0	0	0	0.0000E+00	0.0000E+00	2.00	1.40	0.0000E+00	disp
1	1	4	0	0	4	0	0	0	0	1.0000E+00	1.0000E+00	2.00	1.44	-1.5486E+02	disp

```
Job ends with exit number : 0
```

```
total wall time: 2.00
```

```
total cpu time: 1.47
```

```
exit DEFINITION -----
```

```
= 0 job terminates normally
```

```
= 1 job terminates abnormally (check Fatal Error Message in F06)
```

- **Further information can be found in the f06 file**

Nonlinear Output Control (cont.)

NON - LINEAR I T E R A T I O N S O L U T I O N C O N T R O L P A R A M E T E R S

LOOP CONTROLS FOR : SUBCASE 1, STEP 1, SUBSTEP 0

SOLUTION CONTROL PARAMETERS FROM : NLSTEP ID : 1

Total Time of Loading Case (TOTTIME) 1.00E+00

Maximum Number of Iterations (MAXITER) .. 10

Minimum Number of Iterations (MINITER) .. 1

Maximum Number of Bisection (MAXBIS) 10

Creep Option (CREEP) 0

Number of Fixed Increments (NINC) 1

Interval of Output (NO) 1

Convergence Criteria (CONV) P V

- Displacement (EPSU) -1.00E-01

Tolerance - Residual Force (EPSP) 1.00E-03

- Work (EPSW) 1.00E-01

Option of Rotations and Moments (MRCONV). 3

Matrix Update Option (KMETHOD) PFNT

Matrix Update Increment (KSTEP) 1

Maximum Quasi-Newton Vectors (MAXQN) 0

Maximum Line Searches (MAXLS) 0

Line Search Tolerance (LSTOL) 5.00E-01

Error Tolerance in YF (FSTRESS) 2.00E-01

Standard F06 Output

*** USER INFORMATION MESSAGE 6204 (NL3EMA)

0.000000E+00 SECONDS REQUIRED TO DECOMPOSE MATRIX.

%1.000000E+00	1	1	1.00E+00	2.69E+01	1.00E+00	1.000	1	1	0	4.92E+04	6.533E+06	1.06E+01	-1.587E+02	6	2	1.00	0	0	1
%1.000000E+00	1	2	9.33E-02	1.62E+00	8.19E+02	0.060	0	1	0	2.41E+03	3.233E+06	1.13E+01	-1.558E+02	6	2	1.00	0	0	1
%1.000000E+00	1	3	5.83E-03	1.66E-01	2.07E-02	0.063	0	1	0	1.60E+00	3.233E+06	1.12E+01	-1.549E+02	6	2	1.00	0	0	2
%1.000000E+00	1	4	3.07E-06	7.29E-07	8.63E-03	0.093	0	1	0	2.71E-05	3.233E+06	1.12E+01	-1.549E+02	6	2	1.00	0	0	3

*** JOB CONVERGES FOR THE CURRENT STEP.

*** SUBCASE 1 STEP 1 IS COMPLETED.

Nonlinear Output Control (cont.)

```

LOAD NO.      - - ERROR FACTORS - -   CONV ITR MAT NO.   AVG   TOTL   - - - - - DISP - - - - - LINE_S NO. TOT TOT
STEP INC ITR  DISP      LOAD      WORK  RATE  DIV DIV BIS  R_FORCE WORK  AVG      MAX      AT GRID C FACT NO QNV KUD ITR
%1.00000E+00  1  1 1.00E+00 2.69E+01 1.00E+00 1.000  1  1  0 4.92E+04 6.533E+06 1.06E+01 -1.587E+02      6 2 1.00 0 0 0 0 1

```

- **LOAD STEP**
 - step number minus 1 plus fraction of step, i.e. 0.08 = 8% of first step
- **NO. INC**
 - increment number
- **ITR**
 - iteration number
- **DISP LOAD WORK**
 - displacement, load and energy errors, must be smaller than the tolerances EPSU, EPSP and EPSW
- **CONV RATE**
 - should be between 0 and 1, bigger than 1 means, the solution will never converge

Nonlinear Output Control (cont.)

LOAD	NO.	- - ERROR FACTORS - -				CONV	ITR	MAT	NO.	AVG	TOTL	- - - - - DISP - - - - -				LINE_S	NO.	TOT	TOT		
STEP	INC	ITR	DISP	LOAD	WORK	RATE	DIV	DIV	BIS	R_FORCE	WORK	AVG	MAX	AT	GRID	C	FACT	NO	QNV	KUD	ITR
%1.00000E+00	1	1	1.00E+00	2.69E+01	1.00E+00	1.000	1	1	0	4.92E+04	6.533E+06	1.06E+01	-1.587E+02		6	2	1.00	0	0	0	1

- **ITR DIV**
 - divergence counter, > MAXDIV triggers the divergence process
- **MAT DIV**
 - divergence counter for element and material routines
- **NO. BIS**
 - number of bisections
- **AVG R_FORCE**
 - average residual force (forces and moments); should be small
- **TOTL WORK**
 - approximate total work
- **DISP: AVG MAX AT GRID C**
 - average, maximum displacement at grid in direction c

Nonlinear Output Control (cont.)

LOAD	NO.	- - ERROR FACTORS - -					CONV	ITR	MAT	NO.	AVG	TOTL	- - - - - DISP - - - - -				LINE_S	NO.	TOT	TOT	
STEP	INC	ITR	DISP	LOAD	WORK	RATE	DIV	DIV	BIS	R_FORCE	WORK	AVG	MAX	AT	GRID	C	FACT	NO	QNV	KUD	ITR
%1.00000E+00	1	1	1.00E+00	2.69E+01	1.00E+00	1.000	1	1	0	4.92E+04	6.533E+06	1.06E+01	-1.587E+02		6	2	1.00	0	0	0	1

- **LINE_S: FACT NO**
 - line search factor α and number of line searches
- **NO. QNV**
 - number of Quasi Newton Vectors
- **TOT KUD / ITR**
 - total of stiffness updates / iterations

Nonlinear Output Control (cont.)

- Since V 2010 the new Case Control command NLOPRM is available

Format:

NLOPRM = [OUTCTRL = {STD,SOLUTION,INTERM}]

$$\left[\text{NLDBG} = \left\{ \begin{array}{l} \text{NONE} \\ \text{NLBASIC, NRDBG, ADVDBG, } \left\{ \begin{array}{l} \text{N3DBAS} \\ \text{N3DMED} \\ \text{N3DADV} \end{array} \right\} \end{array} \right\} \right]$$

$$\left[\text{DBGPOST} = \left\{ \begin{array}{l} \text{NONE} \\ \text{LTIME} \\ \text{LSTEP} \\ \text{LSUBC} \\ \text{ALL} \end{array} \right\} \right], \left[\text{MPCPCH} = \left\{ \begin{array}{l} \text{NONE} \\ \text{BEGN, OTIME, STEP} \\ \text{TBEGN, YOTIME, YSTEP} \end{array} \right\} \right]$$

Example(s):

NLOPRM OUTCTRL=STD, SOLUTION DBGPOST=LTIME
 NLOPRM OUTCTRL= (SOLUTION, INTERM) , MPCPCH= (OTIME, STEP)

Nonlinear Output Control (cont.)

- **Example using: nloprm nldbg=advdbg**

```
*** USER INFORMATION MESSAGE 6204 (NL3EMA)
0.000000E+00 SECONDS REQUIRED TO DECOMPOSE MATRIX.
```

```
maximum residual force at node      6 degree of freedom 1 is equal to    5.566E+05
maximum reaction force at node     1 degree of freedom 1 is equal to    2.072E+04
residual convergence ratio      2.687E+01
```

```
maximum residual moment at node     5 degree of freedom 6 is equal to    4.656E+04
maximum reaction moment at node     1 degree of freedom 6 is equal to    9.982E+04
residual convergence ratio      4.664E-01
```

```
maximum displacement change at node  6 degree of freedom 2 is equal to    1.587E+02
maximum displacement increment at node 6 degree of freedom 2 is equal to    1.587E+02
displacement convergence ratio      1.000E+00
```

```
maximum rotation change at node     6 degree of freedom 6 is equal to    2.381E-01
maximum rotation increment at node   6 degree of freedom 6 is equal to    2.381E-01
rotation convergence ratio      1.000E+00
```

```
strain energy change at this iteration is 6.53309E+06
strain energy change at this increment is 6.53309E+06
relative energy error is               1.00000E+00
```

```
%1.00000E+00  1  1 1.00E+00 2.69E+01 1.00E+00 1.000 1  1  0 4.92E+04 6.533E+06 1.06E+01 -1.587E+02      6 2  1.00 0  0  0  1
```

**F06 Output:
First iteration**

Nonlinear Output Control (cont.)

```

0          N O N - L I N E A R   I T E R A T I O N   M O D U L E   S O L U T I O N   D A T A

LOOP CONTROLS :

SUBCASE      1          STEP      1

MAXBIS ..... 10
MAXR ..... 0.1000E+01
RTOLB ..... 0.0000E+00
0 DMAP CONTROL PARAMETERS FROM PREVIOUS ITERATION :

ACCUMULATED TIME ... 0.000E+00
CONVERGENCE ..... YES
NEW STEPCASE ..... YES
NEW MATRIX ..... YES
CURRENT TIME STEP .. 0
LARGE DISPLACEMENT . YES
LAST STIFFNESS UP .. 1
LOOPING LIMIT ..... 10
LANGLE METHOD ..... 3

CORE STATISTICS :          3          75          147          66987          132525          134797 794075017 793675090

PROBLEM STATISTICS :          36          0          0          0          36          6          30          30          0          0

FCB :      1      2      3      4      5      6      7      8      9      10      11      12      13      14      15      16      17
          132597 132705      0      0 132725 132733 132805 132805 133153 133429 133501 133573 133645 133717 133801 133801 133873
          18      19      20      21      22      23      24      25      26      27      28
          29      30      31 134089 134089 134089 134089 134161 134293 134365      0
FCB :      0      0      0 134365 134437 134509 134653

```

F06 Output:
using diag 51

Nonlinear Output Control (cont.)

-
-
-

- Inside the Line Search Loop

NL3LSL: DISPLACEMENT VECTOR (P-SET)

1 :	0.0000D+00	0.0000D+00	0.0000D+00	0.0000D+00	0.0000D+00	0.0000D+00	0.0000D+00	-8.8914D+00	0.0000D+00	0.0000D+00
11 :	0.0000D+00	-8.5714D-02	0.0000D+00	-3.3021D+01	0.0000D+00	0.0000D+00	0.0000D+00	-1.5238D-01	0.0000D+00	-6.8579D+01
21 :	0.0000D+00	0.0000D+00	0.0000D+00	-2.0000D-01	0.0000D+00	-1.1176D+02	0.0000D+00	0.0000D+00	0.0000D+00	-2.2857D-01
31 :	0.0000D+00	-1.5874D+02	0.0000D+00	0.0000D+00	0.0000D+00	-2.3810D-01				

NL3LSL: NONLINEAR INTERNAL FORCE VECTOR (P-SET)

1 :	-2.0717D+04	1.0193D+03	0.0000D+00	0.0000D+00	0.0000D+00	9.9817D+04	-1.3046D+05	1.7284D+04	0.0000D+00	0.0000D+00
11 :	0.0000D+00	-3.8105D+03	-1.7305D+05	3.9327D+04	0.0000D+00	0.0000D+00	0.0000D+00	-1.5121D+04	-1.4869D+05	4.4360D+04
21 :	0.0000D+00	0.0000D+00	0.0000D+00	-3.1892D+04	-8.3709D+04	2.8614D+04	0.0000D+00	0.0000D+00	0.0000D+00	-4.6557D+04
31 :	5.5663D+05	-1.3061D+05	0.0000D+00	0.0000D+00	0.0000D+00	-2.6159D+04				

NL3LSL: LOAD VECTOR (P-SET)

1 :	0.0000D+00	0.0000D+00	0.0000D+00	0.0000D+00	0.0000D+00	0.0000D+00	0.0000D+00	0.0000D+00	0.0000D+00	0.0000D+00
11 :	0.0000D+00	0.0000D+00	0.0000D+00	0.0000D+00	0.0000D+00	0.0000D+00	0.0000D+00	0.0000D+00	0.0000D+00	0.0000D+00
21 :	0.0000D+00	0.0000D+00	0.0000D+00	0.0000D+00	0.0000D+00	0.0000D+00	0.0000D+00	0.0000D+00	0.0000D+00	0.0000D+00
31 :	0.0000D+00	-1.0000D+02	0.0000D+00	0.0000D+00	0.0000D+00	0.0000D+00				

NL3DEL: LOADING ERROR VECTOR(FE-SET)

1 :	1.3046D+05	-1.7284D+04	0.0000D+00	0.0000D+00	0.0000D+00	3.8105D+03	1.7305D+05	-3.9327D+04	0.0000D+00	0.0000D+00
11 :	0.0000D+00	1.5121D+04	1.4869D+05	-4.4360D+04	0.0000D+00	0.0000D+00	0.0000D+00	3.1892D+04	8.3709D+04	-2.8614D+04
21 :	0.0000D+00	0.0000D+00	0.0000D+00	4.6557D+04	-5.5663D+05	1.3051D+05	0.0000D+00	0.0000D+00	0.0000D+00	2.6159D+04

%1.00000E+00 1 1 1.00E+00 2.69E+01 1.00E+00 1.000 1 1 0 4.92E+04 6.533E+06 1.06E+01 -1.587E+02 6 2 1.00 0 0 0 1

Analysis Chaining

- **In SOL 400, all SUBCASEs are independent from each other**
 - Like in SOL 101, many independent subcases can be performed in one single run
 - To change back to the old SOL 106/129 behavior, use system cell 366 (STPFLG)=1
- **For analysis chaining, STEPs have to be defined**
 - A step starts at the end of the step before or the step defined by NLIC
 - Each step can be performed with another type of analysis, i.e. statics, nonlinear transient, normal modes, etc.

Analysis Chaining (cont.)

- **For the ANALYSIS command the following disciplines can be used**

– NLSTATIC	Nonlinear static analysis
– NLTRAN	Nonlinear transient analysis
– STATICS	Linear static analysis
– BUCK	Linear buckling
– MODES	Normal modes analysis
– MFREQ	Modal frequency response
– DFREQ	Direct frequency response
– MTRAN	Modal transient response
– DCEIG	Direct complex eigenvalue analysis
– MCEIG	Modal complex eigenvalue analysis
– BSQUEAL	Break squeal analysis
– HSTAT	Steady state heat transfer analysis
– HTRAN	Transient heat transfer analysis

SPC and MPC Changes

- The SPCs and MPCs are allowed to change from one step to the next for both static and transient analysis

```
.
  NLPARM = 1
STEP 1
  LOAD = 1
  SPC = 1  $ grid 1 fixed, grid 2 free to move
STEP 2
  LOAD = 2
  SPC = 2  $ grid 1 fixed, grid 2 in y is frozen at the location
$          it was at the end of step 1
STEP 3
  ANALYSIS = NLTRAN
  TSTEPNL = 3
  DLOAD = 3
  SPC = 1  $ disp of grid 2 in y is free again
BEGIN BULK $ and starts from its current value.
.
SPC1,1,123456,1  $ no SPC requested on grid 2.
SPC1,2,123456,1  $ same as original constraint for grid 1,
SPC1,2,2,2      $ extra SPC is requested on y dof of grid 2
```

SPC and MPC Changes (cont.)

- **SPC1 is used when a displacement is to be frozen (see page before)**
- **Before MD R3, for SPCs, only absolute boundary changes were allowed using SPCD or SPC**
 - For example, when the displacement is 0.1 at the end of step 1 it will get the value of SPCD or SPC in step 2
- **Starting with MD R3 incremental boundary changes can be done using SPCR (not NLTRAN)**
 - For example, if the displacement is 0.1 at the end of step 1, and there is a 0.3 on an SPCR called in step 2, the actual position will be 0.4
- **MPC changes are only incremental**
 - the relative deformation described with an MPC which is reached at the beginning of a step will be frozen
 - MPC activated in STEP 2: the MPC dof move together in STEP 2
 - STEP 1, stretch/preload a rubber band. STEP 2 use MPC to attach to structure

Restart

- **Restart via Case Control NLRESTART**

```
ASSIGN MASTER='cold.MASTER'  
RESTART  
SOL 400  
CEND  
NLRESTART SUBCASE 1, STEP 2, TIME 0.3    $ before any SUBCASE command  
$  
SUBCASE 1  
    TSTEPNL = 10  
    STEP 1                $ Case control of cold start with STEP 1 and 2  
        LOAD = 10  
    STEP 2                $ Restart begins at time 0.3 of this step  
        TSTEPNL = 30  
        LOAD = 20  
    STEP 3  
        LOAD = 30  
  
BEGIN BULK  
TSTEPNL,30, ,0.01,5  
ENDDATA
```

Restart (cont.)

NLRESTART (SOL 400)

Nonlinear Restart Request

Request a RESTART execution at a specified point for SOL 400.

Format:

$$\text{NLRESTART} \left[\text{SUBCASE } i \left[, \text{STEP } j \left[, \begin{array}{c} \text{LOADFAC } f \\ \text{TIME } t \end{array} \right] \right] \right]$$

Example:

```
NLRESTART SUBCASE 1, STEP 2, LOADFAC 0.3
```

Describer	Meaning
i	Specifies the identification number of a previously executed SUBCASE (Integer, Default is the first SUBCASE).
j	Specifies the identification number of a previously executed STEP (Integer, Default is the first STEP).
f	Specifies the load factor of a previously executed load increment in nonlinear static analysis (Real, $0.0 \leq f \leq 1.0$, Default = 0.0).
t	Specifies the time of a previously executed time step in nonlinear transient analysis (Real, $t_0 \leq t \leq t_n$, where t_0 is the initial time of STEP j, and t_n is the last time of STEP j; Default = t_0).

Introduction to Nonlinear Transient Dynamics

Review of Transient Analysis

- **Static analysis:**

- Compute a solution U that satisfies the equilibrium equation: All follower effects on right hand side.

$$F(U) = P$$

- **Transient analysis:**

- Compute a solution U that satisfies the equilibrium equation:

$$\begin{array}{ccccccc} I(\ddot{U}, t) & + & D(\dot{U}, t) & + & F(U, t) & = & P(t, U) \\ \text{Inertia} & & \text{Damping} & & \text{Element} & & \text{External} \\ \text{Forces} & & \text{Forces} & & \text{Forces} & & \text{Load} \end{array}$$

Review of Transient Analysis (cont.)

- **For a linear system**

$$M\ddot{U} + B\dot{U} + KU = P(t)$$

- **For a general nonlinear system**

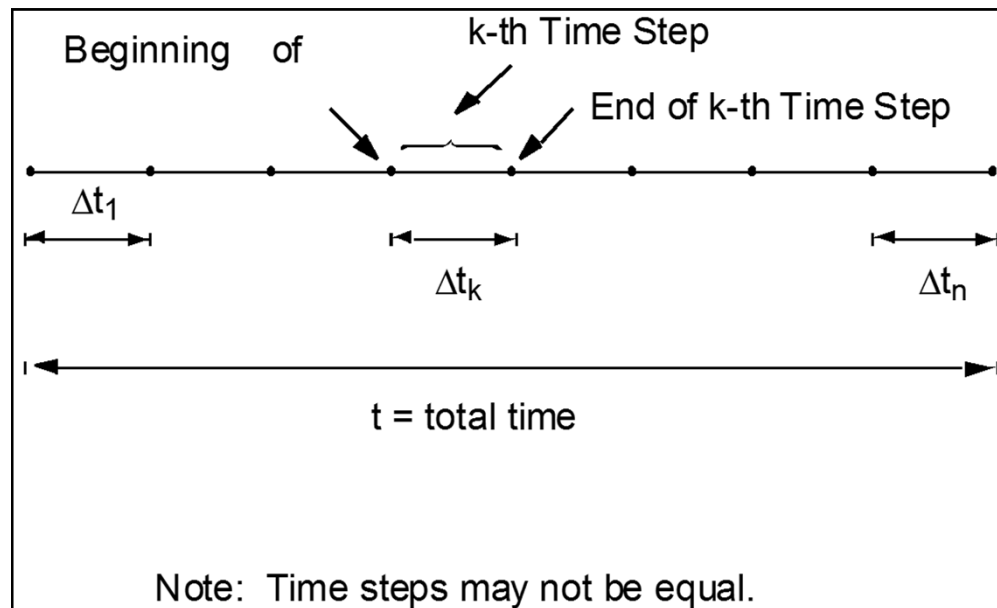
- Mass of the system may change
- Damping may change
- Stiffness may change
- Load may be function of system response
- In MSC Nastran mass and damping cannot change (except for CBUSH1D).
The equilibrium equation is

$$M\ddot{U}(t) + [B\dot{U}(t) + F_{\text{bush1d}}(\dot{U}(t))] + K(U(t)) = P(t, U)$$

Review of Transient Analysis (cont.)

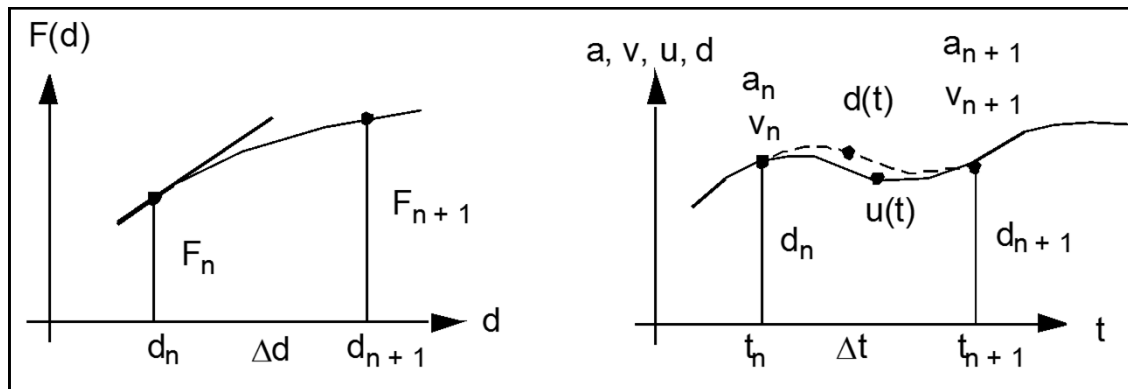
- **Nonlinear Transient Analysis**

- Nonlinear transient analysis proceeds by dividing the time into a number of small time steps. (Finite difference solution)



Review of Transient Analysis (cont.)

- The solution at the end of a time step provides the initial conditions for the next time step.
- For each time step, a relationship is assumed between displacement, velocity, and acceleration (integration scheme).



u_n Displacement at time t_n approximated by d_n .

$\dot{u}_n$ Velocity at time t_n approximated by v_n .

$\ddot{u}_n$ Acceleration at time t_n approximated by a_n .

Review of Transient Analysis (cont.)

- There are a number of different integration schemes available in the literature.
 - Implicit integration: d_{n+1} is obtained by using the equilibrium conditions at time t_{n+1} .
 - Explicit integration: d_{n+1} is obtained by using the equilibrium conditions at time t_n . (SOL 700)
- Use of the integration scheme reduces the transient equilibrium equation to a static equilibrium equation form.

$$\begin{array}{ccc}
 K * (M, B, K, \Delta t) \Delta U & = & P * (\Delta t, \dot{U}, \ddot{U}, M, B, \Delta P) \\
 \text{Effective Dynamics} & & \text{Effective Dynamic} \\
 \text{Stiffness} & & \text{Load Vector}
 \end{array}$$

- Effective dynamic stiffness and load vector depend on the integration scheme used.
- For example, for the average acceleration scheme, also called the trapezoidal rule or Newmark scheme ($\gamma = 1/2$, $\beta = 1/4$)... (see next slide)

Review of Transient Analysis (cont.)

$$\mathbf{K}^* = \frac{4}{\Delta t^2} \mathbf{M} + \frac{2}{\Delta t} \mathbf{B} + \mathbf{K}$$

$$P^* = \Delta P(t) + M[2\ddot{U}(t) + \frac{4}{\Delta t}\dot{U}(t)] + 2B\dot{U}(t)$$

- The equilibrium is satisfied at the beginning and at the end of a time step.
- The equilibrium is not satisfied within the time step. Therefore, the selection of Δt is important.
- A large value of Δt reduces accuracy.
- A small value of Δt increases computing cost.
- A strategy is needed that automatically adjusts the time step value to achieve an optimum value in terms of accuracy and computing cost.
- Adjustment of time step value requires the reformation and decomposition of the dynamic stiffness.

User Interface

- **Solution sequences**
 - SOL 400
- **Solution strategy**
 - TSTEPNL Bulk Data entry.
 - TSTEPNL Case Control command.
 - NLSTEP – New time stepping entry (MD Nastran 2010).
- **Mass specification**
 - RHO field in MATi Bulk Data entries.
 - CMASSi Bulk Data entries for scalar mass elements.
 - CONMi Bulk Data entries for concentrated mass elements.
 - PARAM,COUPMASS, to specify the generation of coupled rather than lumped mass matrices for elements with coupled mass capability.
 - PARAM,WTMASS.

User Interface (cont.)

- **Damping specification**
 - CVISC Bulk Data entry for the viscous damper element
 - Field GE in MATi Bulk Data entries for nonlinear element damping
 - PARAM, G for overall structural damping

User Interface (cont.)

- **Damping specification**

- PARAM, W3 to convert structural damping to equivalent viscous damping
- PARAM, W4 to convert element damping to equivalent viscous damping
- PARAM, NDAMP (.01 default) to specify numerical damping
- Most general damping (non CBUSH1D):

$$B = B^1 + B^{2gg} + \frac{g}{w_3}(K^1 + K^{2gg}) + \frac{1}{w_4}K^4 + \alpha_1 M + \alpha_2(K^1 + K^{2gg}) + BH_{\text{(if component modes)}}$$

$$K^4 = \sum g_e K_e$$

User Interface (cont.)

- **Load Specification**

TLOAD1	Transient load as ordered time, force pairs.
TLOAD2	Transient load as defined by analytical functions.
DAREA	Transient load scale factors.
LSEQ	Generate transient load history for static loads.
NOLIN1	Nonlinear transient load as a tabular function.
NOLIN2	Nonlinear transient load as the product of two variables.
NOLIN3	Nonlinear transient load as a positive variable raised to a power.
NOLIN4	Nonlinear transient load as a negative variable raised to a power.

- Selected by DLOAD, LOADSET, and NONLINEAR Case Control commands.

User Interface (cont.)

- **Initial conditions specification**
 - TIC Bulk Data entry
 - IC Case Control command
- **Additional entries for nonlinear analysis**
 - Similar to nonlinear static analysis
 - Material nonlinear
 - MATS1
 - Geometric nonlinear
 - PARAM,LGDISP,+1

User Interface (cont.)

- **Additional entries for nonlinear analysis (Cont.)**
 - Contact only
 - BCTABLE/BCBODY
 - Combined material and geometric nonlinear
 - MATS1
 - PARAM,LGDISP,+1

Example Input for SOL 400

```

NASTRAN SYSTEM(316)=19
SOL 400
CEND
TITLE = nonlinear static analysis of beam
ECHO=NONE
SUBCASE 1
STEP 1
  ANALYSIS = NLSTATIC
  NLPARM = 1
  SPC = 2
  LOAD = 2
  NLSTRESS(PLOT)=ALL
  DISPLACEMENT(PLOT, SORT1, REAL)=ALL
  SPCFORCES(PLOT, SORT1, REAL)=ALL
  STRESS(PLOT, SORT1, REAL, VONMISES, BILIN)=ALL
BEGIN BULK
PARAM      POST      0
PARAM      PRTMAXIM YES
PARAM      IGDISP     1
NLPARM     1          10          PFNT     1          25          UPW      YES
$ Elements and Element Properties for region : beam
PSOLID     1          1          0
$ Pset: "beam" will be imported as: "psolid.1"
CHEXA      1          1          1          2          103          102          506          507
           608          607
CHEXA      2          1          2          3          104          103          507          508
           609          608
.....
MAT1       1          3.+7          .3          7.4-4
$ Nodes of the Entire Model
GRID       1          0.          0.          0.
GRID       2          1.          0.          0.
GRID       3          2.          0.          0.
.....
$ Loads for Load Case : Default
SPCADD     2          1
LOAD       2          1.          1.          1
$ Displacement Constraints of Load Set : fixed
SPC1       1          123          1          102          203          304          405          506
           607          708          809          910
$ Nodal Forces of Load Set : tip_load
FORCE      1          101          0          1500.          0.          -1.          0.
FORCE      1          505          0          1500.          0.          -1.          0.
FORCE      1          606          0          1500.          0.          -1.          0.
FORCE      1          1010          0          1500.          0.          -1.          0.
$ Referenced Coordinate Frames
ENDDATA 4c78738d

```


General Features

- **Transient material nonlinear, geometric nonlinear, combined geometric and material nonlinear, and contact problems can be solved using this solution sequence**
- **Linear superelements can be combined with nonlinear elements**
- **Component Modal reduction (SEQSET,EIGR) available for the linear superelements**

Integration Schemes

- **Two-Point Integration Scheme**

- Use the following equilibrium equation:

$$M \ddot{U}_{n+1} + B \dot{U}_{n+1} + F_{n+1} = P_{n+1}$$

- Assume that the acceleration for a time step is equal to the average of the beginning and end of the step.

$$\ddot{U}(t) = \frac{\ddot{U}_n + \ddot{U}_{n+1}}{2}$$

- Velocity and displacement are obtained by integration.

$$\dot{U}_{n+1} = \dot{U}_n + \left(\frac{\ddot{U}_n + \ddot{U}_{n+1}}{2} \right) \Delta t$$

$$U_{n+1} = U_n + \dot{U}_n \Delta t + \left(\frac{\ddot{U}_n + \ddot{U}_{n+1}}{4} \right) \Delta t^2$$

Integration Schemes (cont.)

- Rearrange the equilibrium equation in terms of incremental values.

$$\left[\frac{4}{\Delta t^2} M + \frac{2}{\Delta t} B + K_T \right] \Delta U = P_{n+1} + P_n - F_n + \frac{4}{\Delta t} M \dot{U}_n - \left[\frac{4}{\Delta t^2} M + \frac{2}{\Delta t} C \right] [U_{n+1} - U_n]$$

Dynamic Stiffness Dynamic Load Factor

- Calculate velocity as follows:

$$\dot{U}_{n+1} = (U_{n+1} - U_n) \frac{2}{\Delta t} - \dot{U}_n$$

- Note that the acceleration need not be calculated since it does not appear in the incremental equilibrium equation.
- For postprocessing purposes, acceleration is calculated as:

$$\{\ddot{U}_n\} = \frac{1}{\Delta t_n + \Delta t_{n+1}} \left[\frac{\Delta t_n}{\Delta t_{n+1}} \{\ddot{U}_{n-1}\} + \left(\frac{\Delta t_{n+1}}{\Delta t_n} - \frac{\Delta t_n}{\Delta t_{n+1}} \right) \{\dot{U}_n\} - \frac{\Delta t_{n+1}}{\Delta t_n} \{\ddot{U}_{n-1}\} \right]$$

TSTEPNL for Nonlinear Transient

1	2	3	4	5	6	7	8	9	10
TSTEPNL	ID	NDT	DT	NO	METHOD	KSTEP	MAXITER	CONV	
	EPSU	EPSP	EPSW	MAXDIV	MAXQN	MAXLS	FSTRESS		
	MAXBIS	ADJUST	MSTEP	RB	MAXR	UTOL	RTOLB	MINITER	

- HHT (Houbolt, Hughes, and Taylor) time integration scheme is used. It has better numerical damping than the Newmark-Beta.
- NDAMP (for B, F and P) set to 0.0 (i.e., Default, for contact)
- NDAMPM (for M) set to 1.0 (i.e., Default, for contact)

The above default settings for NDAMP, and NDAMPM results in use of the Single Step Houbolt Operator, and is recommended for problems with contact.

- NDAMP (for B, F and P) set to -0.05 (i.e., Default, for non-contact)
- NDAMPM (for M) set to 0.0 (i.e., Default, for non-contact)

M, B, F, and P are respectively the mass, damping, internal force, and external load.

Generalized-Alpha Scheme

- **Existing HHT scheme has been extended to Generalized-Alpha Scheme**

- Generalized-Alpha Scheme is a two-parameter scheme that allows spectral radius to vary between 0.0 and 1.

$$M\ddot{u}_{n+1} + \alpha_m C\dot{u}_{n+1} + F_{n+1}^{int} = F_{n+1}^{ext}$$

- No Special input needed to flag new scheme
 - For contact / impact problems, SOL400 by default uses $\alpha_f = 0.0$, $\alpha_m = 1.0$ (this corresponds to spectral radius of 0.0).
 - For non-contact problems, SOL400 continues to use $\alpha_f = -0.05$, $\alpha_m = 0.0$ (this ensures back-ward compatibility)

Generalized-Alpha Scheme (cont.)

- α_f and α_m can be manually changed by user via PARAM,NDAMP,xxx and PARAM,NDAMPPM,yyy respectively in the Case Control or Bulk Data sections.
- Valid range of α_f is $[-0.5, 0.0]$. Valid range of α_m is $[0.0, 1.0]$. A variety of time-stepping schemes are obtained by varying α_f and α_m

• $\alpha_f = 0.0$	$\alpha_m = 0.0$	Newmark-Beta
• $\alpha_f = -0.05$	$\alpha_m = 0.0$	HHT (Default for non-contact)
• $\alpha_f = 0.0$	$\alpha_m = 1.0$	Single-Step Houbolt (Default for contact)

TSTEPNL – Time Step and Results

1	2	3	4	5	6	7	8	9	10
TSTEPNL	ID	NDT	DT	NO					

- **DT, NDT – time increment and how many**
- **NO – number of increments after which output is desired**
 - if positive, the output locations are controlled by DT. Default=1.
 - if negative, output will be performed after NO actual computed increments (SOL 129 style)

TSTEPNL – Results Output

- **Additional User Control over Output using OTIME. Example:**

```
OTIME = 99
SET 99 = 0.025, 0.035 → additional output
$
SUBCASE 10
  STEP 1
    LOAD = 1
    NLPARM = 110
  STEP 2
    ANALYSIS = NLTRAN
    DLOAD = 3
    TSTEPNL = 130
BEGIN BULK
$. . . . . 2 . . . . . 3 . . . . . 4 . . . . . 5 . . . . . 6 . . . . . 7 . . . . . 8 . . . . . 9 . . . . . 0
TSTEPNL 130      100      0.01      10      AUTO
```

↑
“NO” field, output for $t = 0, 1 \cdot 10 \cdot 0.01, 2 \cdot 10 \cdot 0.01, \dots$

TSTEPNL – Results Output (cont.)

- **The Data grouping option allows to minimize disk space usage (especially important for Transient because of many increments).**
 - PARAM, NLPACK, n
 - N=-1 – output data as requested, but the restart data are stored only for the end of a STEP. So, restart is possible from end of step only.
 - N=1 – each package of data on the database includes the output data for one output time step and the corresponding restart data. Restart can be done from each output time step.

TSTEPNL – Stiffness Update

1	2	3	4	5	6	7	8	9	10
TSTEPNL	ID					METHOD	KSTEP		
						MAXQN	MAXLS		
								MINITER	

- **METHOD (default=AUTO, for contact: FNT)**
 - ADAPT – Automatic adjustment of the time steps. Stiffness updates for every KSTEPth converged bisection solution.
 - AUTO, SEMI – Automatic stiffness updates to improve convergence. Automatic adjustment of time steps. SEMI = stiffness update after the first iteration.
 - ITER – Stiffness update at every KSTEPth iteration. Automatic adjustment of time steps.

TSTEPNL – Stiffness Update (cont.)

- FNT – Full Newton iteration. Stiffness update at every iteration. Defaults of EPSU, EPSP and EPSW = 0.01 and MAXLS=2. Default for contact.
- PFNT – Pure Full Newton iteration. Same as FNT except that the defaults for EPSU and EPSW = -0.01, EPSP=0.01 and MAXLS=0.
- For FNT and PFNT, whether the stiffness will be updated upon convergence and the start of the next increment, depends on KSTEP field:
 - KSTEP=“BLANK” – decided by SOL 400 based on element type. Default.
 - KSTEP= 1 – stiffness will not be updated
 - KSTEP=-1 – stiffness will always be updated

TSTEPNL – Stiffness Update (cont.)

1	2	3	4	5	6	7	8	9	10
TSTEPNL	ID					METHOD	KSTEP		
						MAXQN	MAXLS		
								MINITER	

- **MAXQN (Default=10)**
 - number of Quasi Newton Vectors for which an exact stiffness update takes place. Default=10.
- **MAXLS (Default=2)**
 - maximum number of line searches. For PFNT, Default=0.
- **MINITER (Default=1)**
 - minimum number of iterations. Default for Contact=2.

TSTEPNL – Convergence

1	2	3	4	5	6	7	8	9	10
TSTEPNL	ID							CONV	
	EPSU	EPSP	EPSW						

- **Same as NLPARM**

TSTEPNL – Divergence

1	2	3	4	5	6	7	8	9	10
TSTEPNL	ID							MAXITER	
				MAXDIV				FSTRESS	
	MAXBIS				MAXR			RTOLB	

- **Same as NLPARM**

TSTEPNL – Adaptive Time Stepping

1	2	3	4	5	6	7	8	9	10
TSTEPNL	ID	NDT	DT						
		ADJUST	MSTEP	RB	MAXR	UTOL			

- Implemented for METHOD=ADAPT, AUTO, SEMI, ITER, FNT and PFNT
- Controlled via ADJUST and related parameters
- Time step will be adjusted to $T / MSTEP$ whereby T is the dominant period of response

$$\lambda_n = \frac{\Delta \mathbf{u}_n^T \mathbf{K} \Delta \mathbf{u}_n}{\Delta \mathbf{u}_n^T \mathbf{M} \Delta \mathbf{u}_n} \rightarrow T = \frac{2\pi}{\text{sqrt}(\lambda)}$$

TSTEPNL – Adaptive Time Stepping (cont.)

1	2	3	4	5	6	7	8	9	10
TSTEPNL	ID	NDT	DT						
		ADJUST	MSTEP	RB	MAXR	UTOL			

- To prevent thrashing a stepping function is being applied

$$\Delta t_{n+1} = f(r) \Delta t_n$$

with

$f = 0.25$	for	$r < 0.5 * RB$
$f = 0.5$	for	$0.5 < RB \leq r < RB$
$f = 1.0$	for	$RB \leq r < 2.0$
$f = 2.0$	for	$2.0 \leq r < 3.0/RB$
$f = 4.0$	for	$r \geq 3.0/RB$

Bounds on Δt :

$$\text{MIN}\left(\frac{DT}{2^{\text{MAXBIS}}}, \frac{DT}{\text{MAXR}}\right) < \Delta t < \text{MAXR} * DT$$

TSTEPNL – Adaptive Time Stepping (cont.)

1	2	3	4	5	6	7	8	9	10
TSTEPNL	ID	NDT	DT						
		ADJUST	MSTEP	RB	MAXR	UTOL			

- **UTOL, tolerance on velocities**
If the velocities are low time step will not be adjusted
- **To switch Adaptive Time Stepping off, just set ADJUST to zero**

1	2	3	4	5	6	7	8	9	10
TSTEPNL	ID	NDT	DT	NO	METHOD	KSTEP	MAXITER	CONV	
	EPSU	EPSP	EPSW	MAXDIV	MAXQN	MAXLS	FSTRESS		
	MAXBIS	ADJUST	MSTEP	RB	MAXR	UTOL	RTOLB	MINITER	

TSTEPNL – Recommendations

- For the advanced nonlinear elements and materials and Contact Analysis FNT or PFNT with CONV=PV should be used. Default.
- In all other cases (excluding GAPS) AUTO and CONV=UPW can be used. For GAPS use ITER with KSTEP=1.
- For contact with friction FNT and UPW has been found to be more precise
- The PFNT method is more stringent than the FNT. However if CONV=UPW is used, both methods give similar results.

Nonlinear Output

```

0                                N O N - L I N E A R   I T E R A T I O N   M O D U L E   O U T P U T

      STIFFNESS UPDATE TIME      0.01 SECONDS                      SUBCASE      1      STEP      1
      ITERATION TIME            0.14 SECONDS

      - TIME STEP -      - - ERROR FACTORS - -      CONV ITR MAT  AVG      TOTL      - - - - - DISP - - - - - NO. TOT TOT
      TIME      NO. BIS ADJUST ITR  DISP      LOAD      WORK      RATE DIV DIV R_FORCE  WORK      AVG      MAX      AT GRID C QNV KUD ITR

%1.00000E-04      1  0 1.0000  2 8.19E-04 2.75E-04 2.66E-03  0.00  0  1  1.4E-05 1.467E-01 1.38E-03  5.818E-02      1421 1  1  0  2
%2.00000E-04      2  0 1.0000  4 8.78E-04 6.09E-03 2.62E-03  0.34  0  1  2.0E-04 2.939E+00 8.10E-03  3.158E-01      1421 1  4  0  6
*** USER INFORMATION MESSAGE 7600 (NL3LOP)
      THE INPUT TIME STEP SIZE 1.00000D-04 IS USED AS THE INITIAL TIME STEP SIZE.
*** SYSTEM INFORMATION MESSAGE 6916 (MREORDR)
      DECOMP ORDERING METHOD CHOSEN: DEFAULT, ORDERING METHOD USED: BEND

%1.00000E-04      1  0 1.0000  2 8.19E-04 2.75E-04 2.66E-03  0.00  0  1  1.4E-05 1.467E-01 1.38E-03  5.818E-02      1421 1  1  0  2
%2.00000E-04      2  0 1.0000  4 8.78E-04 6.09E-03 2.62E-03  0.34  0  1  2.0E-04 2.939E+00 8.10E-03  3.158E-01      1421 1  4  0  6
%3.00000E-04      3  0 1.0000  7 2.19E-04 8.66E-03 8.50E-04  0.49  0  1  8.3E-04 2.170E+01 2.48E-02  9.055E-01      1441 1 10  0 13
%4.00000E-04      4  0 1.0000 11 1.82E-03 4.78E-01 2.07E-03  0.81  0  1  4.1E-02 9.711E+01 5.35E-02  1.871E+00      1441 1 20  0 24
*** USER INFORMATION MESSAGE 4550 (nl3con)
*** THE STIFFNESS MATRIX IS UPDATING TO THE CURRENT SOLUTION ***

*** JOB DOES NOT CONVERGE AT THE CURRENT TIME STEP OR INCREMENT.

%4.00000E-04      4  0 1.0000 13 1.16E-02 4.68E+00 1.10E-02  0.91  2  1  1.9E-01 9.720E+01 5.34E-02  1.871E+00      1441 1  1  1  26
*** USER INFORMATION MESSAGE 6191 (nl3con)
*** DIVERGING SOLUTION SEQUENCE INITIATED ***
*** USER INFORMATION MESSAGE 4550 (nl3con)
*** THE STIFFNESS MATRIX IS UPDATING TO THE PREVIOUS ITERATION ***

*** JOB DOES NOT CONVERGE AT THE CURRENT TIME STEP OR INCREMENT.

%4.00000E-04      4  0 1.0000 15 3.82E-04 9.49E-03 3.67E-04  0.06  0  1  3.9E-04 9.813E+01 5.38E-02  1.889E+00      1441 1  1  2  28
*** USER INFORMATION MESSAGE 6191 (nl3con)
*** SOLUTION CONVERGED, DIVERGING SOLUTION SEQUENCE ENDS ***

```

F06 Output

Nonlinear Output (cont.)

0																				NON - LINEAR I T E R A T I O N M O D U L E O U T P U T															
STIFFNESS UPDATE TIME					0.01 SECONDS										SUBCASE		1		STEP		1														
ITERATION TIME					0.14 SECONDS																														
- TIME STEP -					- - ERROR FACTORS - -					CONV ITR MAT		AVG		TOTL		- - - - - DISP - - - - -			NO. TOT TOT																
TIME		NO. BIS		ADJUST		ITR		DISP		LOAD		WORK		RATE		DIV		DIV		R_FORCE		WORK		AVG		MAX		AT GRID C		QNV		KUD		ITR	
%1.00000E-04		1		0		1.0000		2		8.19E-04		2.75E-04		2.66E-03		0.00		0		1		1.4E-05		1.467E-01		1.38E-03		5.818E-02		1421		1 1 0 2			

- TIME
 - current time
- TIME STEP NO.
 - time step number. Initialized to 0 at the beginning of each STEP.
- TIME STEP BIS
 - number of bisections performed
- TIME STEP ADJUST
 - ratio of the current time increment to the original DT
- TIME STEP ITR
 - number of iterations within the time increment

Nonlinear Output (cont.)

0	NON - LINEAR I T E R A T I O N M O D U L E O U T P U T																
STIFFNESS UPDATE TIME		0.01 SECONDS				SUBCASE				1		STEP		1			
ITERATION TIME		0.14 SECONDS															
- TIME STEP -		- - ERROR FACTORS - -				CONV ITR MAT		AVG		TOTL		- - - - - DISP - - - - -		NO. TOT TOT			
TIME	NO. BIS ADJUST ITR	DISP	LOAD	WORK	RATE	DIV	DIV	R_FORCE	WORK	AVG	MAX	AT GRID C	QNV	KUD	ITR		
%1.00000E-04	1 0 1.0000 2	8.19E-04	2.75E-04	2.66E-03	0.00	0	1	1.4E-05	1.467E-01	1.38E-03	5.818E-02	1421	1	1	0	2	

- DISP LOAD WORK
 - displacement, load and energy errors, must be smaller than the tolerances EPSU, EPSP and EPSW
- CONV RATE
 - should be between 0 and 1, bigger than 1 means the solution will never converge
- ITR DIV
 - divergence counter, > MAXDIV triggers the divergence process
- MAT DIV
 - divergence counter for element and material routines

Nonlinear Output (cont.)

0																			NON - LINEAR I T E R A T I O N M O D U L E O U T P U T																				
STIFFNESS UPDATE TIME										0.01 SECONDS					SUBCASE					1		STEP			1														
ITERATION TIME										0.14 SECONDS																													
- TIME STEP -																			- - ERROR FACTORS - -				CONV ITR MAT		AVG TOTL - - - - - DISP - - - - - NO. TOT TOT														
TIME		NO.		BIS		ADJUST		ITR		DISP		LOAD		WORK		RATE		DIV		DIV		R_FORCE		WORK		AVG		MAX		AT GRID C		QNV		KUD		ITR			
%1.00000E-04		1		0		1.0000		2		8.19E-04		2.75E-04		2.66E-03		0.00		0		1		1.4E-05		1.467E-01		1.38E-03		5.818E-02		1421		1		1		0		2	

- **AVG R_FORCE**
 - average residual force, should be small
- **TOTL WORK**
 - approximate total energy
- **DISP: AVG MAX AT GRID C**
 - average, maximum displacement at grid in direction c
- **NO. QNV**
 - Number of Quasi Newton Vectors
- **TOT KUD / ITR**
 - Total of stiffness updates / iterations

Mass Specification

- Similar to linear transient analysis.
- CMASS1 and CMASS2 define scalar mass elements.
- CMASS3 and CMASS4 define scalar mass elements connected only to scalar points.
- CONM1 defines a 6 x 6 mass matrix for a grid point.
- CONM2 defines a diagonal mass matrix for translational degrees of freedom and a 3 x 3 full matrix for rotational degrees of freedom at a grid point.
- Element mass density is defined on the RHO field of the MATi Bulk Data entry.
- PARAM,COUPMASS,1 specifies the coupled mass matrix for elements with coupled mass capability (BAR, BEAM, ROD, HEXA, PENTA, TRIA, and TUBE elements).
- Non structural mass PSHELL, PBEAM, etc.

Damping

- **Damping represents energy dissipation observed in structures.**
- **Difficult to accurately model since damping results from many mechanisms:**
 - Viscous effects (dashpot, shock absorber)
 - External friction (slippage in structural joints)
 - Internal friction (characteristic of material type)
 - Structural nonlinearities (plasticity)
- **Analytical conveniences are used to model damping.**
- **Viscous damping force proportional to velocity**

$$f_v = b\dot{u}$$

$$m\ddot{u} + b\dot{u} + ku = p$$

Damping (cont.)

- **Structural damping force proportional to displacement**

$$f_s = i G k u \qquad i = \sqrt{-1}$$

$$m\ddot{u} + (1 + iG)ku = p \qquad G = \text{structural damping coefficient}$$

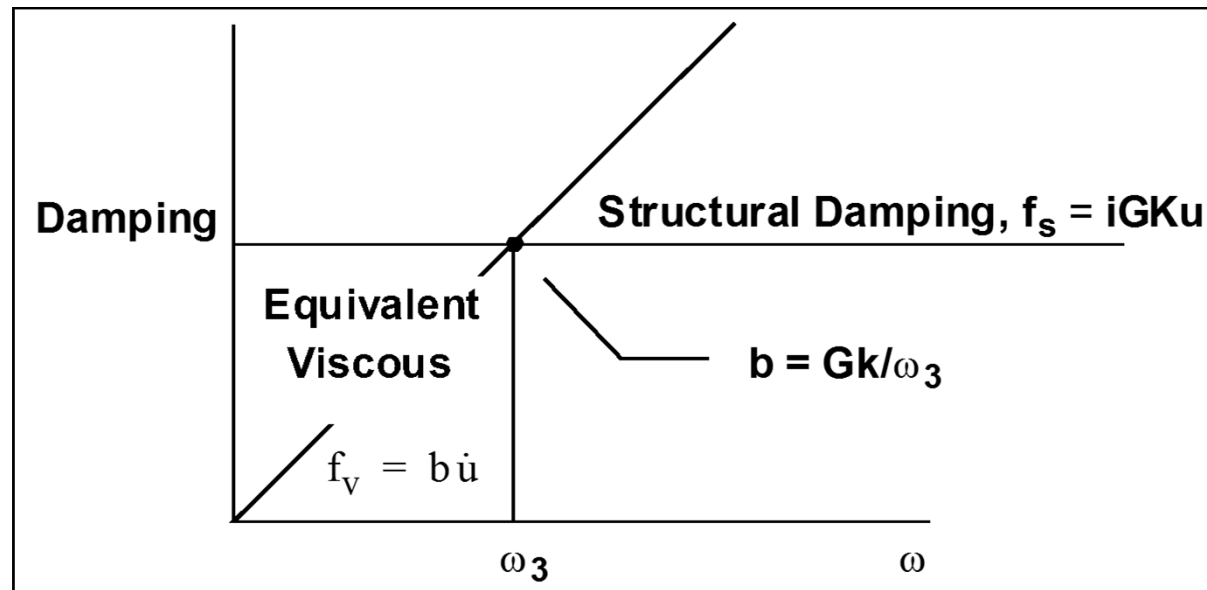
- **Viscous and structural damping are equivalent at frequency ω_3 .**

with

$$G = \frac{b\omega_3}{k} = \frac{2\xi\omega_3}{\omega_n}$$

$$\xi = \frac{c}{2m\omega_n}$$

Damping (cont.)



Damping (cont.)

- **Similar to linear transient analysis.**
- **Damping matrix **B** comprised of several matrices:**

$$B = B^1 + B^{2gg} + \frac{g}{w_3}(K^1 + K^{2gg}) + \frac{1}{w_4} \sum g_e K_e + F_{CBUSH1D}(\dot{U}, t)$$

- Where
 - B^1 = damping elements (VISC,DAMP)
 - G = overall structural damping coefficient (PARAM,G)
 - W_3 = frequency of interest - rad/sec (PARAM,W3)
 - K^1 = global stiffness matrix
 - G_e = element structural damping coefficient (GE on the MATi entry)
 - W^4 = frequency of interest - rad/sec (PARAM,W4)
 - K_e = element stiffness matrix

Damping (cont.)

- **Default values for W3 and W4 are 0.0. In this case, the associated damping terms are ignored.**
- **Element damping is provided with PARAM,W4 and field GE in the MATi entry.**
- **Damping matrix is not rotated.**
 - Caution for large rotation.

Damping (cont.)

- **DAMPING PARAMETERS**

- PARAM,G, factor (default = 0.0)
 - Overall structural damping coefficient to multiply stiffness matrix for linear elements.
- PARAM,W3, factor (default = 0.0)
 - Converts overall structural damping to equivalent viscous damping.
- PARAM,W4 factor (default = 0.0)
 - Converts element structural damping to equivalent viscous damping.
- Units for W3,W4 are radians/unit time.
- If PARAM,G is used; PARAM,W3 must be set to greater than zero or PARAM,G will be ignored.

Load Specification

- **Three ways:**
 - Dynamic loads
 - Static loads
 - Nonlinear loads

Dynamic Loads

- **Dynamic loads require both temporal and spatial distribution.**
- **A user needs to follow four steps to specify dynamic loads.**
- **The four steps are:**
 1. Define the load as a function of time (TLOADi).
 2. Define the spatial distribution of the load (DAREA, FORCE, etc.).
 3. Combine the TLOADi entries via DLOAD entry.
 4. Select the loads via the DLOAD Case Control command.

Dynamic Loads (cont.)

- **TLOAD1 Bulk Data Entry**

- Description: Defines a time-dependent dynamic load or enforced motion of the form

$$\{P(t)\} = \{A * F(t - \tau)\}$$

for use in transient response analysis.

- Format:

1	2	3	4	5	6	7	8	9	10
TLOAD1	SID	EXCITEID	DELAYI/ DELAYR	TYPE	TID/F	USO	VSO		

- Example:

TLOAD1	5	7	15	LOAD	13				

Dynamic Loads (cont.)

<u>Field</u>	<u>Contents</u>
SID	Set identification number. (Integer > 0).
EXCITEID	Identification number of static load, DAREA, SPCD or thermal load set
DELAYI	Identification number of DELAY Bulk Data entry that defines time delay τ . (Integer > 0 or blank)
DELAYR	Value of time delay τ that will be used for all degrees-of-freedom that are excited by this dynamic load entry (Real or blank)
TYPE	Defines the nature of the dynamic excitation. (Integer; character or blank; Default = 0)
TID/F	Identification number of TABLEDi entry that gives $F(t)$. (Integer > 0)
USO	Factor for initial displacements of the enforced DOF
VSO	Factor for initial velocities of the enforced DOF

Dynamic Loads (cont.)

- **DAREA Bulk Data Entry**

- Description: Defines scale (area) factors for dynamic loads. DAREA is used in conjunction with RLOADi and TLOADi entries.
- Format:

1	2	3	4	5	6	7	8	9	10
DAREA	SID	P1	C1	A1	P2	C2	A2		

- Example:

DAREA	3	6	2	8.2	15	1	10.1		

<u>Field</u>	<u>Contents</u>
SID	Identification number. (Integer > 0).
Pi	Grid, extra, or scalar point identification number.(Integer > 0).
Ci	Component number. (Integer 1 through 6 for grid point; blank or 0 for extra or scalar point).
Ai	Scale (area) factor. (Real).

Dynamic Loads (cont.)

- TLOAD2 Bulk Data Entry**

- Description: Defines a time-dependent dynamic load or enforced motion of the form

$$\{P(t)\} = \begin{cases} 0 & , t < (T1 + \tau) \text{ or } t > (T2 + \tau) \\ \{A\} \tilde{t}^B e^{C\tilde{t}} \cos(2\pi F\tilde{t} + P) & , (T1 + \tau) \leq t \leq (T2 + \tau) \end{cases}$$

for use in a transient response problem where $\tilde{t} = t - T1 - \tau$.

- Format:

1	2	3	4	5	6	7	8	9	10
TLOAD2	SID	EXCITEID	DELAYI/ DELAYR	TYPE	T1	T2	F	P	
	C	B	USO	VSO					

- Example:

TLOAD2	4	10			2.1	4.7	12		
	2								

Dynamic Loads (cont.)

<u>Field</u>	<u>Contents</u>
SID	Set identification number. (Integer > 0).
EXCITEID	Identification number of static load, DAREA, SPCD, or thermal load set (in heat transfer analysis) that defines A. (Integer > 0).
DELAYI	Identification number of DELAY Bulk Data entry that defines time delay τ . (Integer > 0 or blank)
DELAYR	Value of time delay τ that will be used for all degrees-of-freedom that are excited by this dynamic load entry (Real or blank)
TYPE	Defines the nature of the dynamic excitation. (Integer; character or blank; Default = 0)
T1	Time constant. (Real ≥ 0.0).
T2	Time constant. (Real; $T2 > T1$).
F	Frequency in cycles per unit time. (Real ≥ 0.0 ; Default = 0.0).
P	Phase angle in degrees. (Real; Default = 0.0).
C	Exponential coefficient. (Real; Default = 0.0).
B	Growth coefficient. (Real; Default = 0.0).
USO	Factor for initial displacements of the enforced DOF
VSO	Factor for initial velocities of the enforced DOF

Dynamic Loads (cont.)

- **For a constant load, leave fields F, P, C, and B blank.**
- **For a cosine wave, specify $F = 1.0$, and leave fields P, C, and B blank.**
- **For a sine wave, specify $F = 1.0$, $P = -90^\circ$ and leave fields C and B blank.**

Dynamic Loads (cont.)

- **DLOAD Bulk Data Entry**

- Description: Defines a dynamic loading condition for frequency response or transient response problems as a linear combination of load sets defined via RLOAD1 or RLOAD2 entries for frequency response or TLOAD1 or TLOAD2 entries for transient response.

- Format:

1	2	3	4	5	6	7	8	9	10
DLOAD	SID	S	S1	L1	S2	L2	S3	L3	
	S4	L4							

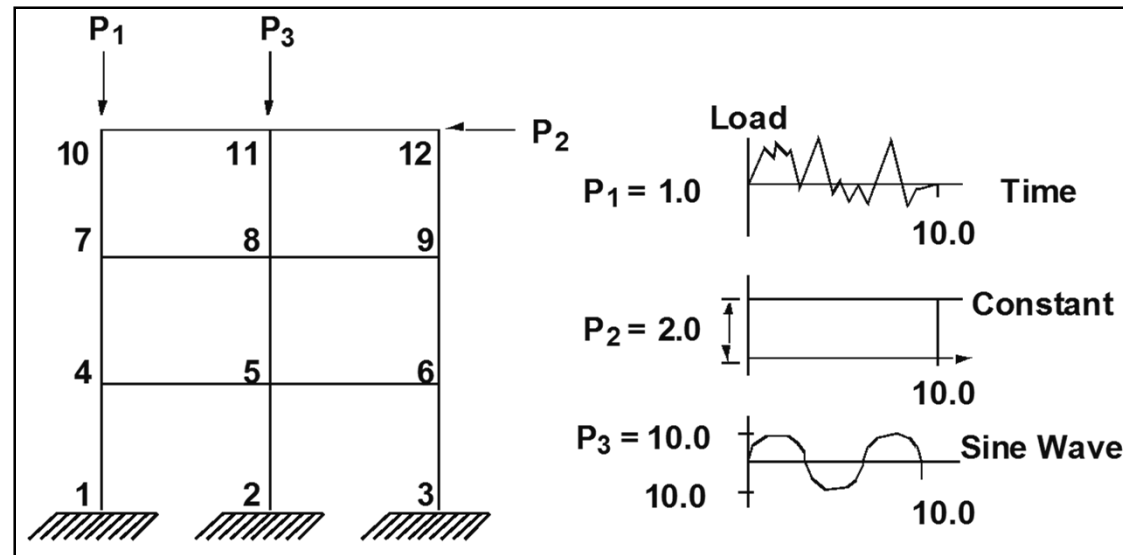
- Example:

DLOAD	17	1.	2.	6	-2.	7	2.	8	
	-2.	9							

Dynamic Loads (cont.)

<u>Field</u>	<u>Contents</u>
SID	Load set identification number. (Integer > 0).
S	Scale factor. (Real).
Si	Scale Factors. (Real).
Li	Load set identification numbers of RLOAD1, RLOAD2, TLOAD1, and TLOAD2 entries. (Integer > 0).

Dynamic Loads Example



- **Step 1**
 - P1 : TLOAD1,101,1,0,0,1
 - P2 : TLOAD1,102,2,0,0,2
 - or
TLOAD2,102,2,0,0.0,10.0
 - P3 : TLOAD2,103,3,,0,0.0,10.0,1.0,-90.0

Dynamic Loads Example (cont.)

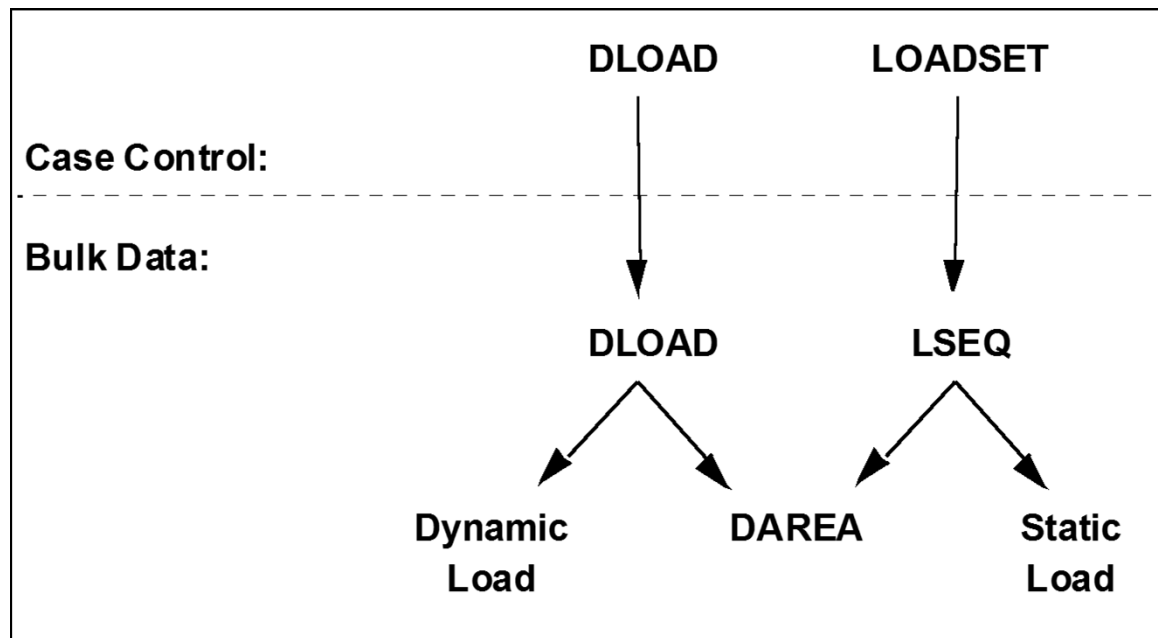
- **Step 2**
 - DAREA,1,10,2,-1.0
 - DAREA,2,12,1,-2.0
 - DAREA,3,11,2,-10.0
- **Step 3**
 - DLOAD,10,1.0,1.0,101,1.0,102,1.0,103
- **Step 4**
 - DLOAD=10 in Case Control

Static Loads in Transient Analysis

- **A user needs to follow five steps to specify static loads in transient analysis.**
- **The five steps are:**
 1. Define the static loads using FORCEi, GRAV, MOMENTi, etc., that are referenced by the LOAD Case Control command.
 2. Define a LSEQ Bulk Data entry to point to a TLOADi entry and to a load set that is referenced by a LOADSET Case Control command.
 3. Define a TLOAD1 or TLOAD2 entry to define a constant function with time.
 4. Combine all the TLOADi entries through the DLOAD Bulk Data entry.
 5. Select the DLOAD entry through the DLOAD Case Control command and the LSEQ entry through the LOADSET Case Control command.

LSEQ Entry

- Defines static loads that will be applied dynamically.
- Relationship to other commands and entries:



LSEQ Entry (cont.)

- **LSEQ Bulk Data Entry**

- Description: Defines a sequence of static load sets.

- Format:

1	2	3	4	5	6	7	8	9	10
LSEQ	SID	EXCITEID	LID	TID					

- Example:

LSEQ	100	200	1000	1001					

Field

Contents

SID

Set identification of the set of LSEQ entries. (Integer > 0).

EXCITEID

The EXCITEID set identification assigned to this static load vector. (Integer > 0).

LSEQ Entry (cont.)

<u>Field</u>	<u>Contents</u>
LID	Load set identification number of a set of static load entries such as those Referenced by the LOAD Case Control command. (Integer > 0 or blank).
TID	Temperature set identification of a set of thermal load entries such as those referenced by the TEMP(LOAD) Case Control Command. (Integer > 0 or blank).

Example: Static Loads in Transient Analysis

- **Aim:** to specify gravity load in transient analysis.
- **Solution:**

Case Control Section

- Step 5: DLOAD = 50011
LOADSET = 5000 → to LSEQ

Bulk Data Section

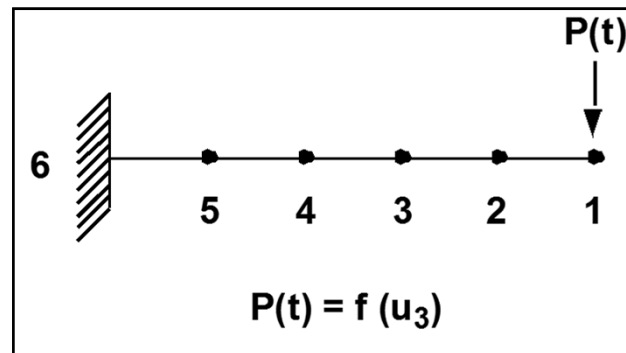
- Step 4: DLOAD, 50011, 1.0, 1.0, 5001, 1.0, 4444,....
Additional Loads
- Step 3: TLOAD2, 5001, 5002, , 0, 0.0, 99999., 0., 0.

Example: Static Loads in Transient Analysis (cont.)

- Step 3: TLOAD2, 5001, 5002, , 0, 0.0, 99999., 0., 0. Defines a cos function
DAREA Link ID
- Step 2: LSEQ, 5000, 5002, 5555
LOADSET ID
- Step 1: GRAV, 5555, , 380., 0., 0., 1.0

Nonlinear Loads

- Allows for the specification of load at a particular degree of freedom to be the function of displacement and velocity at another degree of freedom.
- Example:



- Load at grid point 1, displacement component 2 as a function of the displacement component 1 at grid point 3.

Nonlinear Loads (cont.)

- **Useful for specifying nonlinear springs and nonlinear damping.**
- **Nonlinear loads are specified using NOLINi entries.**
- **Four NOLINi entries (NOLIN1, NOLIN2, NOLIN3, and NOLIN4) to specify mechanical loads.**
- **Nonlinear loads are selected via the NONLINEAR Case Control command.**
- **Output of nonlinear loads via NLLOAD Case Control command.**
- **Nonlinear loads cannot be selected via the DLOAD Case Control command.**
- **All degrees of freedom referenced on NOLINi entry must be members of the solution set.**

Nonlinear Loads (cont.)

- **Velocity for an independent degree of freedom (for the purpose of loads) is calculated as**

$$\dot{U}_t = \frac{U_t - U_{t-1}}{\Delta_t}$$

Note: This may be different from that calculated in the integration scheme. But it is acceptable.

- **In all NOLINi entries a degree of freedom is specified by the grid number and its component number.**
- **All loads generated with NOLINi entries lag behind by one time step Δt .**

Nonlinear Loads (cont.)

- **NOLIN1 Bulk Data Entry**

- Description: Defines nonlinear transient forcing functions of the form.

Function of displacement: $P_i(t) = S * T(u_j(t))$ (1)

Function of velocity: $P_i(t) = S * T(u_j(t))$ (2)

where $u_j(t)$ and $\dot{u}_j(t)$ are the displacement and velocity at point GJ in the direction of CJ.

- Format:

1	2	3	4	5	6	7	8	9	10
NOLIN1	SID	GI	CI	S	GJ	CJ	TID		

- Example:

NOLIN1	21	3	4	2.1	3	10	6		

Nonlinear Loads (cont.)

<u>Field</u>	<u>Contents</u>
SID	Nonlinear load set identification number. (Integer > 0).
GI	Grid, scalar, or extra point identification number at which nonlinear load is to be applied. (Integer > 0).
CI	Component number for GI. ($0 < \text{Integer} \leq 6$; blank or zero if GI is a scalar or extra point).
S	Scale factor. (Real).
GJ	Grid, scalar, or extra point identification number. (Integer > 0).
CJ	Component number for GJ according to the following table:

Type of point	Displacement	Velocity
Grid	$1 \leq \text{Integer} \leq 6$	$11 \leq \text{Integer} \leq 16$
Scalar	Blank or Zero	Integer = 10
Extra	Blank or Zero	Integer = 10

TID	Identification number of a TABLEDi entry. (Integer > 0).

Nonlinear Loads (cont.)

- **NOLIN2 Bulk Data Entry**

- Description: Defines nonlinear transient forcing functions of the form.

$$P_i(t) = S * X_j(t) * X_k(t)$$

where X_j and X_k can be either displacement or velocity at points G_j and G_k in the directions of C_j and C_k .

- Format:

1	2	3	4	5	6	7	8	9	10
NOLIN2	SID	GI	CI	S	GJ	CJ	GK	CK	

- Example:

NOLIN2	14	2	1	2.9	2	1	2		

Nonlinear Loads (cont.)

- **NOLIN3 Bulk Data Entry**

- Description: Defines nonlinear transient forcing functions of the form.

$$P_i(t) = \begin{cases} S^*[X_j(t)]^A, & X_j(t) > 0 \\ 0 & , X_j(t) \leq 0 \end{cases}$$

where may be a displacement or a velocity at point GJ in the direction of CJ.

- Format:

1	2	3	4	5	6	7	8	9	10
NOLIN3	SID	GI	CI	S	GJ	CJ	A		

- Example:

NOLIN3	4	102		-6.1	2	15	-3.5		

Nonlinear Loads (cont.)

- NOLIN4 Bulk Data Entry**

- Description: Defines nonlinear transient forcing functions of the form.

$$P_i(t) = \begin{cases} -S^*[-X_j(t)]^A, & X_j(t) < 0 \\ 0 & , X_j(t) \geq 0 \end{cases}$$

where X_j may be a displacement or a velocity at point GJ in the direction of CJ.

- Format:

1	2	3	4	5	6	7	8	9	10
NOLIN4	SID	GI	CI	S	GJ	CJ	A		

- Example:

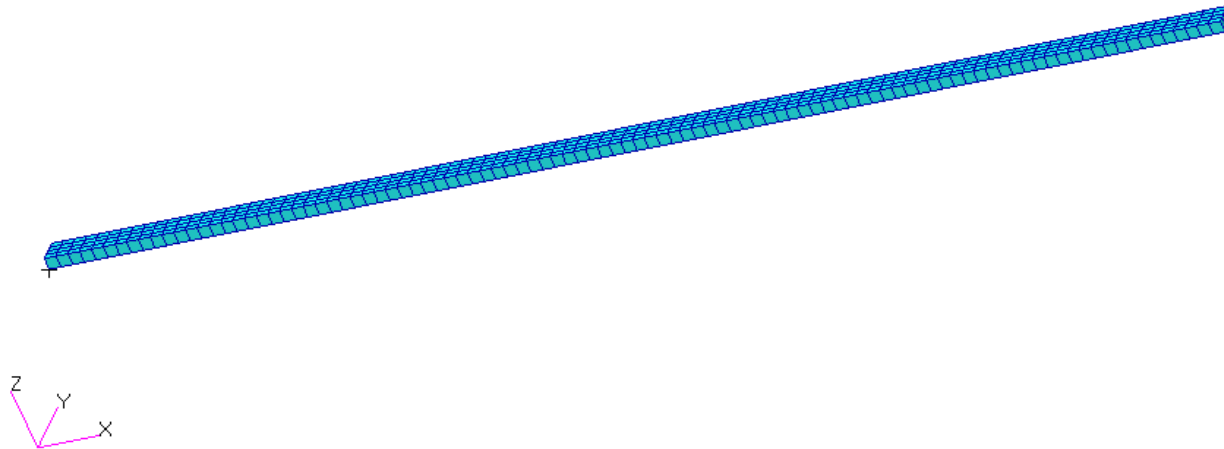
NOLIN4	2	4	6	2	101		16.3		

Initial Conditions

- **May impose initial displacements and/or velocities with a TIC Bulk Data entry.**
- **IC Case Control command selects TIC entries in the Bulk Data Section.**
- **Warning: Initial conditions for unspecified degrees of freedom are set to zero.**
- **Initial conditions may be specified only for A-set degrees of freedom.**

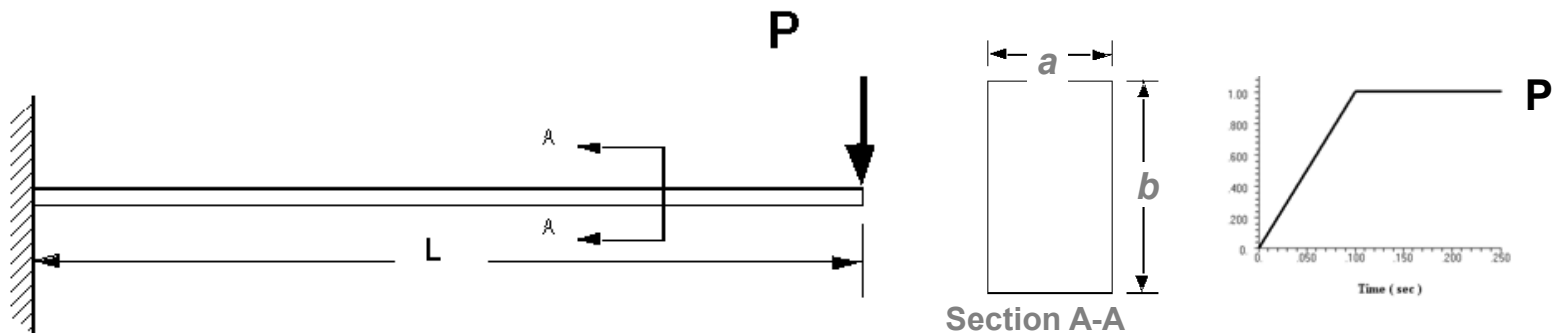
Case Study

Nonlinear Transient Dynamic Analysis of a Beam



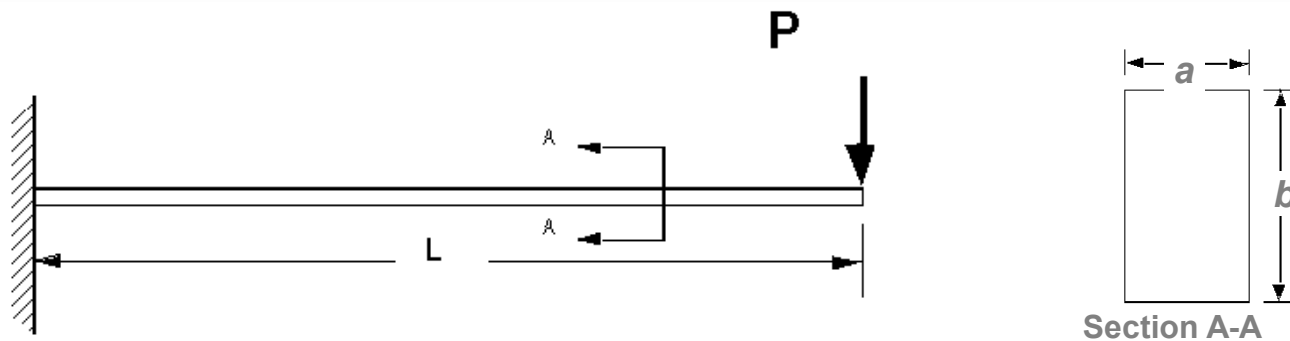
Case Study - Problem Description

- In this study, a cantilever beam is subjected to a ramped load. We will analyze it with a nonlinear transient dynamic procedure, assuming the load is ramped up to the full value of 6000 lbs in 0.10 second.



(Data on next page)

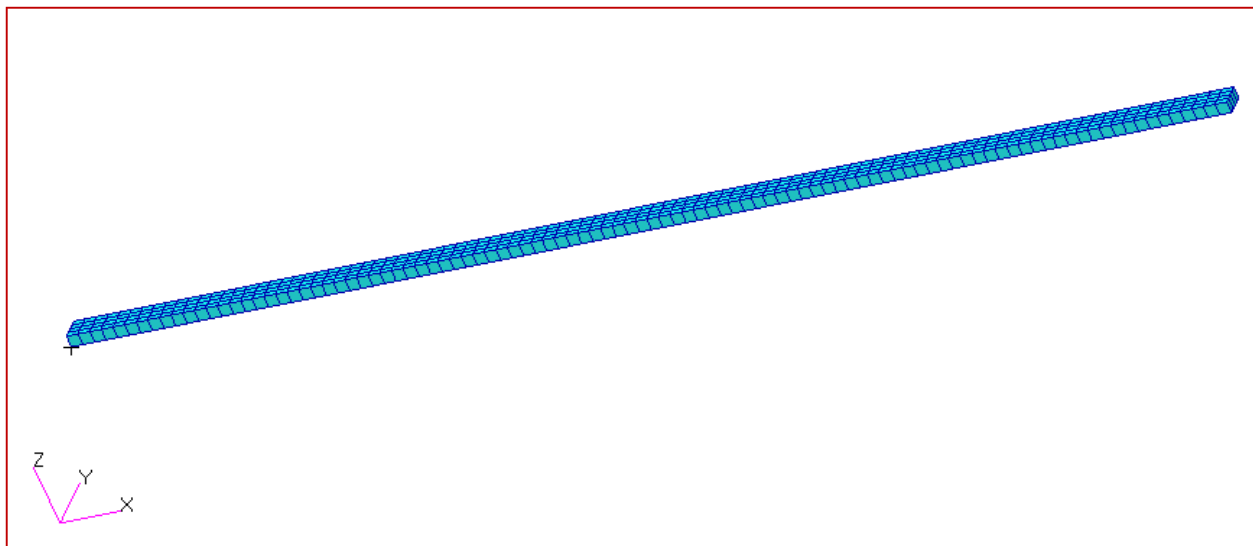
Case Study - Problem Description (cont.)



Length, L	100.0 in	2.54 m
a	1.0 in	25.4 mm
b	2.0 in	50.8 mm
Young's Modulus	$30.0 \times 10^6 \text{ lb/in}^3$	207 GPa
Poisson's Ratio	0.3	0.3
P	6000 lb	27200 N

Case Study - Problem Description (cont.)

- The simulation is to use 3D solid finite elements. The elements are uniformly spaced along the length of the beam (i.e. a mesh 100 elements long and four elements deep).



- Objective:
 - Nonlinear Static vs. Nonlinear Transient Dynamic Analysis.

Case Study - Problem Description (cont.)

- Beginning of the Nastran input file for each run

Nonlinear Static

```

NASTRAN SYSTEM(316)=19
SOL 400
CEND
TITLE = nonlinear static analysis of beam
ECHO=NONE
SUBCASE 1
STEP 1
  ANALYSIS = NLSTATIC
  NLPARM = 1
  SPC = 2
  LOAD = 2
  NLSTRESS (PLOT)=ALL
  DISPLACEMENT (PLOT, SORT1, REAL)=ALL
  SPCFORCES (PLOT, SORT1, REAL)=ALL
  STRESS (PLOT, SORT1, REAL, VONMISES, BILIN)=ALL
BEGIN BULK
PARAM      POST      0
PARAM      PRTMAXIM YES
PARAM      LGDISP     1
NLPARM     1         10          PFNT     1         25      UPW      YES
  
```

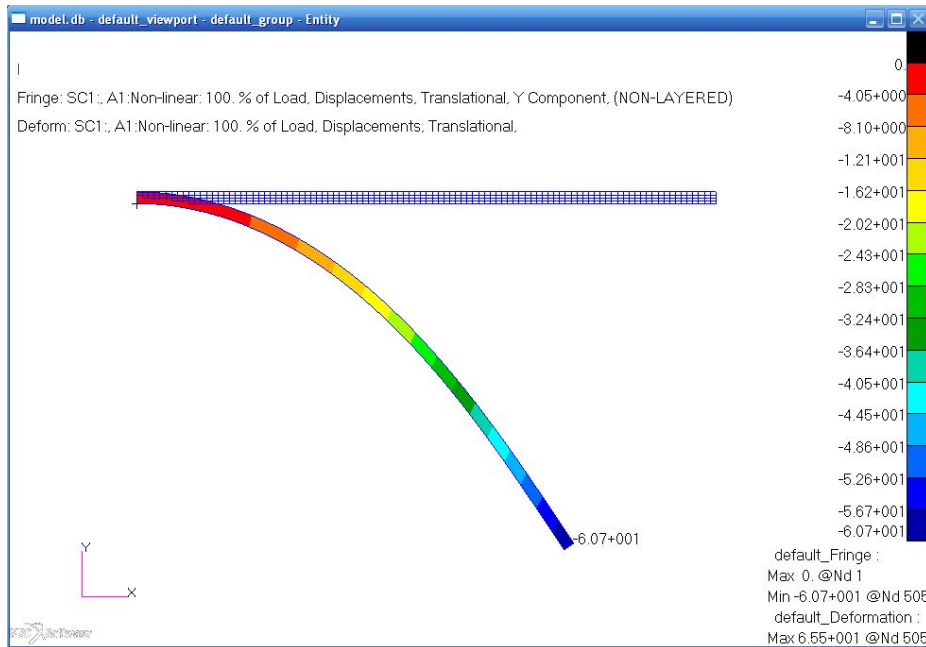
```

NASTRAN SYSTEM(316)=19
SOL 400
CEND
TITLE = nonlinear transient analysis of beam, damped
LOADSET = 1
SUBCASE 1
STEP 1
  ANALYSIS = NLTRAN
  TSTEPNL = 1
  SPC = 2
  DLOAD = 2
  DISPLACEMENT (SORT1, REAL)=ALL
  SPCFORCES (SORT1, REAL)=ALL
  STRESS (SORT1, REAL, VONMISES, BILIN)=ALL
BEGIN BULK
PARAM, G, 0.06
PARAM, W3, 1.0
PARAM      POST      0
PARAM      PRTMAXIM YES
PARAM      LGDISP     1
TSTEPNL     1         250      1.-3          PFNT     1          UPW
  
```

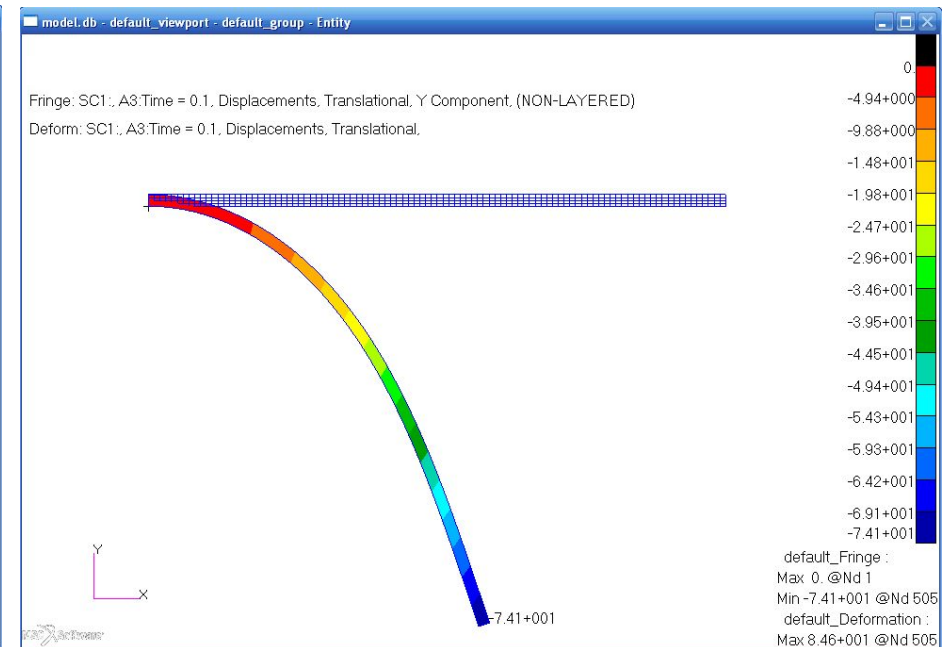
Nonlinear Transient

Case Study - Compare the Results

Y-Displacement at Tip

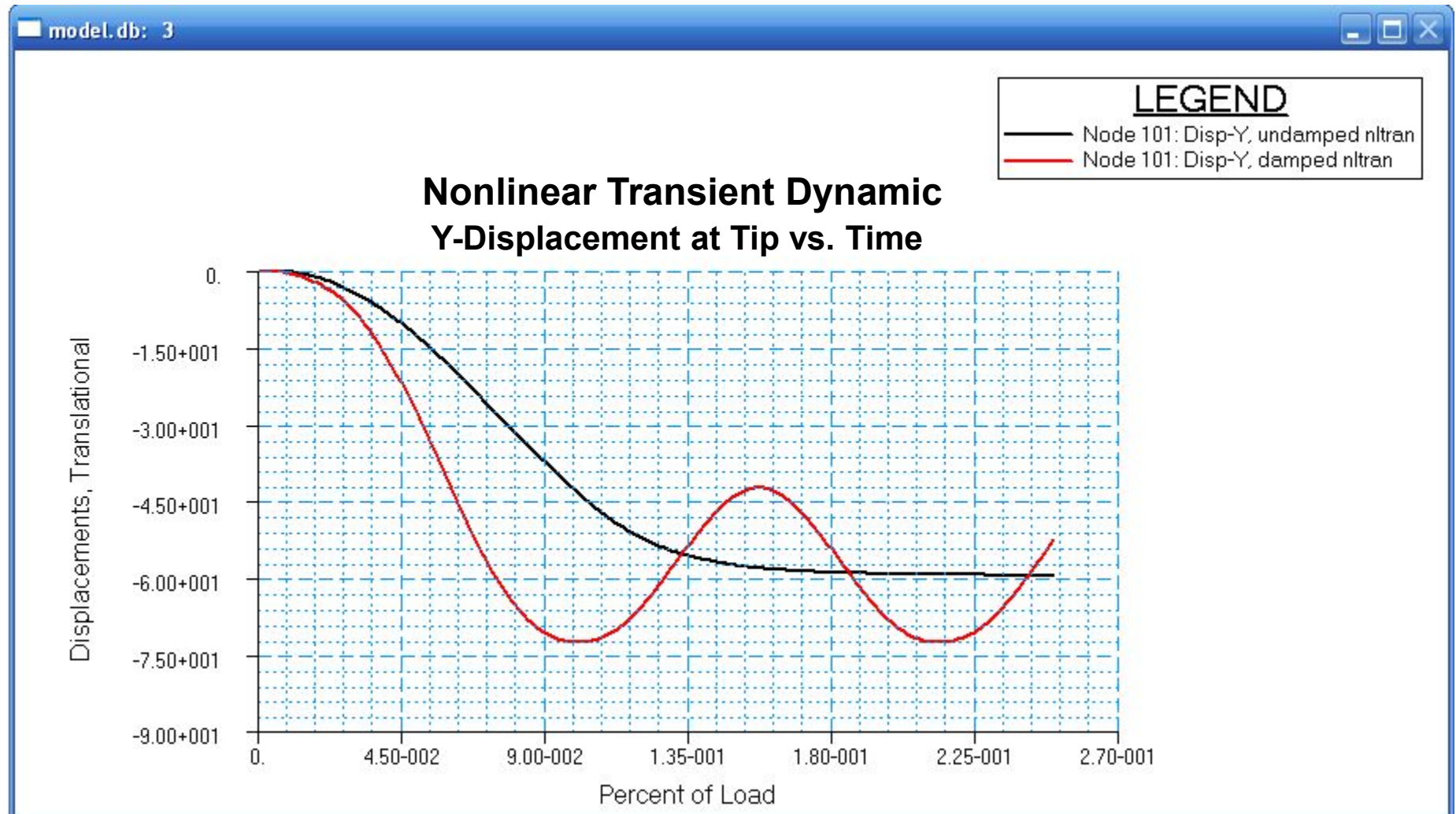


Nonlinear Static



**Nonlinear Transient Dynamic
Undamped (at Time = 0.1)**

Case Study - Compare the Results (cont.)

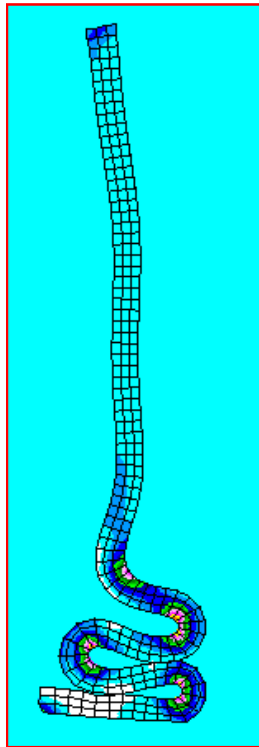


Hints and Recommendations

- **Identify the type of nonlinearity**
- **Localize nonlinear region**
- **Divide time history by subcases for convenience**
- **Select default values to start - TSTEPNL**
- **Pick time step size for highest frequency of interest (twelve or more steps per cycle and frequent content of input)**
- **Some damping is desirable for numerical stability**
- **Avoid massless degrees of freedom**
- **Use PFNT for highly nonlinear problems**

Workshop 6

- Perform Workshop 6: Dynamic Collapse of a Cylinder



SECTION 11

Additional Special Topics in Dynamics

Additional Special Topics in Dynamics

- **There are many classes available for MSC NASTRAN, some of which are very specific to an analysis capability (nonlinear, design optimization, acoustics, etc)**
- **This section will highlight some of these capabilities as they pertain to dynamics**
- **For a full, detailed presentation of these topics, the regular class is available**
 - A list of the classes this section addresses is on the next slide

Additional Special Topics in Dynamics (cont.)

- **Superelements**
 - NAS106A – Basic Substructure Analysis (Primary Superelements)
 - NAS106B – Advanced Substructure Analysis (Secondary Superelements)
- **Rotodynamics**
 - NAS127 – Rotordynamics
- **Acoustics**
 - NAS115 – Fluid Structure Analysis
- **Design Optimization**
 - NAS107 – Design Sensitivity and Optimization
- **Monitor Points**
- **ADAMs MNF creation**

SECTION 12

Superelements in Normal Modes

Normal Modes Analysis Using Parts Superelements

What is a Superelement?

- **Physical and mathematical representation**
 - Physical - substructure: a finite element model of a portion of a structure
 - Mathematical - boundary matrices: loads, mass, damping, and stiffness reduced from the interior points to the exterior or boundary points
- **Other types of substructuring analysis**
 - Cyclic symmetry analysis
 - GENEL and DMIG input
- **There are two ways to define superelements**
 - PARTS superelement (discussed in this section)
 - Main Bulk Data superelement (See Superelement seminar)

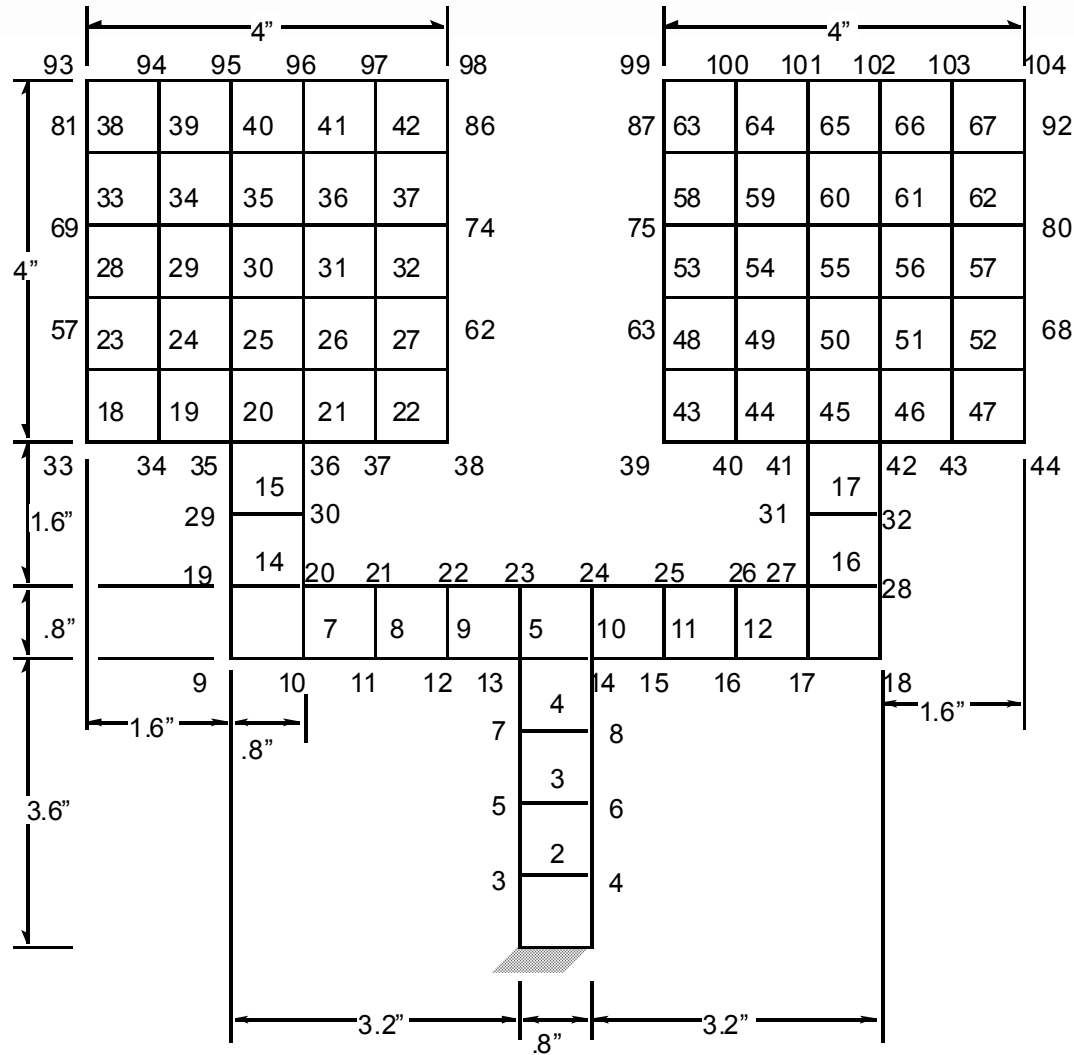
Parts Superelements Defined in MSC Nastran

- The description for each superelement is completely self-contained within its own Bulk Data Section.
- This type of superelement is often referred to as parts in this section.
- Each superelement begins with a
BEGIN [BULK] SUPER=m
delimiter and terminates with another
BEGIN [BULK] SUPER=n
or **ENDATA** delimiter
- The Main Bulk Data Section contains the full description of the residual structure and other superelements not described by parts. These superelements can be defined by either the **SESET** or **SEELT** entries. See Superelement Seminar for further description of **SESET**.

Parts Superelements Defined in MSC Nastran (cont.)

- By default, if the location of a grid point in one part falls within a specified tolerance of a grid point in another part, then these parts will be connected together at that location.
- You can manually overwrite the connection tolerance.
- Duplicate IDs (e.g., grid points, elements, properties, etc.) are allowed across different parts.
- Each part can have its own loadings and component modes calculation.
- Plotting of complete system modes is supported.
- A superelement may also be defined as a copy of a superelement or obtained from outside MSC Nastran.

Sample Problem – Steel Stamping



SAMPLE PROBLEM - STEEL STAMPING (Cont.)

- Grid Points 1 and 2 fixed
- Material properties:

Steel	t	=	.05"
	E	=	29 x 10 ⁶ psi
	n	=	.3
	r	=	.283 lb/in ³ (weight density)

Model Definition for Sample Problem

```
$ file - part-se-guyan.dat
$
$ all 7 s.e. brought in using begin super
$ duplicate boundary grids id
$ each s.e. contains its own property description with the same id
$
$ -----
SOL 103
CEND
TITLE = S.E. SAMPLE PROBLEM 1
SUBTITLE = Normal Modes - Guyan Reduction
DISP = ALL
PARAM,GRDPNT,1
param,post,-1
$
$ use expanded Case Control
$
SUBCASE 1
  LABEL = SE1
  SUPER=1
SUBCASE 2
  LABEL = SE2
  SUPER=2
SUBCASE 3
  LABEL = SE3
  SUPER=3
SUBCASE 4
  LABEL = SE4
  SUPER=4
SUBCASE 5
  LABEL = SE5
  SUPER=5
SUBCASE 6
  LABEL = SE6
  SUPER=6
SUBCASE 7
  LABEL = SE7
  SUPER=7
  SPC=1
SUBCASE 100
  LABEL = Residual Modes
  SUPER=0
  METHOD=100
$
include 'plot.blk'
$
BEGIN BULK
eigr1,100,,,10
```

```
$
$ Residual (SE 0) model
CQUAD4 5      1      13      14      24      23
$
GRID 13      -.4      3.6      0.
GRID 14      .4      3.6      0.
GRID 23      -.4      4.4      0.
GRID 24      .4      4.4      0.
$
include 'prop1.blk'
$
begin super=1
$
include 'prop1.blk'
include 'se1.blk'
$
begin super=2
$
include 'prop1.blk'
include 'se2.blk'
$
begin super=3
$
include 'prop1.blk'
include 'se3.blk'
$
begin super=4
$
include 'prop1.blk'
include 'se4.blk'
$
begin super=5
$
include 'prop1.blk'
include 'se5.blk'
$
begin super=6
$
include 'prop1.blk'
include 'se6.blk'
$
begin super=7
$
include 'prop1.blk'
include 'se7.blk'
$
enddata
```

Model Definition for Sample Problem (cont.)

```
$
$   plot.blk
$
output(plot)
$
set 1 = all
axes z,x,y
view 0.,0.,0.
seupplot 0
ptitle = full structure
find scale, origin 1, set 1
plot set 1 origin 1 label both
$
$   deform plots
$
seplot 1
ptitle = superelement 1
find scale, origin 1, set 1
plot static deformation set 1 origin 1 label both
$
seplot 2
ptitle = superelement 2
find scale, origin 1, set 1
plot static deformation set 1 origin 1 label both
$
seplot 3
ptitle = superelement 3
find scale, origin 1, set 1
plot static deformation set 1 origin 1 label both
$
seplot 4
ptitle = superelement 4
find scale, origin 1, set 1
plot static deformation set 1 origin 1 label both
$
seplot 5
ptitle = superelement 5
find scale, origin 1, set 1
plot static deformation set 1 origin 1 label both
```

```
$
seplot 6
ptitle = superelement 6
find scale, origin 1, set 1
plot static deformation set 1 origin 1 label both
$
seplot 7
ptitle = superelement 7
find scale, origin 1, set 1
plot static deformation set 1 origin 1 label both

$
seplot 0
ptitle = superelement 0
find scale, origin 1, set 1
plot static deformation set 1 origin 1 label both
$
seupplot 0
ptitle = full structure
axes x,mz,y
find scale, origin 1, set 1
plot static deformation set 1 origin 1
$
```

```
$
$   prop1.blk
$
MAT1,1,30.+6,,.3,.283
PARAM,WTMASS,.00259
PARAM,AUTOSPC,YES
PSHELL,1,1,.05,1,,1
$
```


Model Definition for Sample Problem (cont.)

```

$
$      sel.blk
$
CQUAD4 18      1      33      34      46      45
CQUAD4 19      1      34      35      47      46
CQUAD4 20      1      35      36      48      47
CQUAD4 21      1      36      37      49      48
CQUAD4 22      1      37      38      50      49
CQUAD4 23      1      45      46      58      57
CQUAD4 24      1      46      47      59      58
CQUAD4 25      1      47      48      60      59
CQUAD4 26      1      48      49      61      60
CQUAD4 27      1      49      50      62      61
CQUAD4 28      1      57      58      70      69
CQUAD4 29      1      58      59      71      70
CQUAD4 30      1      59      60      72      71
CQUAD4 31      1      60      61      73      72
CQUAD4 32      1      61      62      74      73
CQUAD4 33      1      69      70      82      81
CQUAD4 34      1      70      71      83      82
CQUAD4 35      1      71      72      84      83
CQUAD4 36      1      72      73      85      84
CQUAD4 37      1      73      74      86      85
CQUAD4 38      1      81      82      94      93
CQUAD4 39      1      82      83      95      94
CQUAD4 40      1      83      84      96      95
CQUAD4 41      1      84      85      97      96
CQUAD4 42      1      85      86      98      97
$
GRDSET                                6
$
6
$      boundary grids
$
GRID    35      -3.6      6.      0.
GRID    36      -2.8      6.      0.
$

```

```

GRID    33      -5.2      6.      0.
GRID    34      -4.4      6.      0.
GRID    37      -2.      6.      0.
GRID    38      -1.2      6.      0.
GRID    45      -5.2      6.8      0.
GRID    46      -4.4      6.8      0.
GRID    47      -3.6      6.8      0.
GRID    48      -2.8      6.8      0.
GRID    49      -2.      6.8      0.
GRID    50      -1.2      6.8      0.
GRID    57      -5.2      7.6      0.
GRID    58      -4.4      7.6      0.
GRID    59      -3.6      7.6      0.
GRID    60      -2.8      7.6      0.
GRID    61      -2.      7.6      0.
GRID    62      -1.2      7.6      0.
GRID    69      -5.2      8.4      0.

GRID    70      -4.4      8.4      0.
GRID    71      -3.6      8.4      0.
GRID    72      -2.8      8.4      0.
GRID    73      -2.      8.4      0.
GRID    74      -1.2      8.4      0.
GRID    81      -5.2      9.2      0.
GRID    82      -4.4      9.2      0.
GRID    83      -3.6      9.2      0.
GRID    84      -2.8      9.2      0.
GRID    85      -2.      9.2      0.
GRID    86      -1.2      9.2      0.
GRID    93      -5.2      10.      0.
GRID    94      -4.4      10.      0.
GRID    95      -3.6      10.      0.
GRID    96      -2.8      10.      0.
GRID    97      -2.      10.      0.
GRID    98      -1.2      10.      0.

```

Model Definition for Sample Problem (cont.)

```

$
$   se2.blk
$
CQUAD4  43      1      39      40      52      51
CQUAD4  44      1      40      41      53      52
CQUAD4  45      1      41      42      54      53
CQUAD4  46      1      42      43      55      54
CQUAD4  47      1      43      44      56      55
CQUAD4  48      1      51      52      64      63
CQUAD4  49      1      52      53      65      64
CQUAD4  50      1      53      54      66      65
CQUAD4  51      1      54      55      67      66
CQUAD4  52      1      55      56      68      67
CQUAD4  53      1      63      64      76      75
CQUAD4  54      1      64      65      77      76
CQUAD4  55      1      65      66      78      77
CQUAD4  56      1      66      67      79      78
CQUAD4  57      1      67      68      80      79
CQUAD4  58      1      75      76      88      87
CQUAD4  59      1      76      77      89      88
CQUAD4  60      1      77      78      90      89
CQUAD4  61      1      78      79      91      90
CQUAD4  62      1      79      80      92      91
CQUAD4  63      1      87      88      100     99
CQUAD4  64      1      88      89      101     100
CQUAD4  65      1      89      90      102     101
CQUAD4  66      1      90      91      103     102
CQUAD4  67      1      91      92      104     103
$
GRDSET
$
$   boundary grids
$
GRID    41      2.8      6.      0.
GRID    42      3.6      6.      0.
$

```

6

```

$
GRID    39      1.2      6.      0.
GRID    40      2.      6.      0.
GRID    43      4.4      6.      0.
GRID    44      5.2      6.      0.
$
GRID    51      1.2      6.8      0.
GRID    52      2.      6.8      0.
GRID    53      2.8      6.8      0.
GRID    54      3.6      6.8      0.
GRID    55      4.4      6.8      0.
GRID    56      5.2      6.8      0.
GRID    63      1.2      7.6      0.
GRID    64      2.      7.6      0.
GRID    65      2.8      7.6      0.
GRID    66      3.6      7.6      0.
GRID    67      4.4      7.6      0.
GRID    68      5.2      7.6      0.
GRID    75      1.2      8.4      0.

GRID    76      2.      8.4      0.
GRID    77      2.8      8.4      0.
GRID    78      3.6      8.4      0.
GRID    79      4.4      8.4      0.
GRID    80      5.2      8.4      0.
GRID    87      1.2      9.2      0.
GRID    88      2.      9.2      0.
GRID    89      2.8      9.2      0.
GRID    90      3.6      9.2      0.
GRID    91      4.4      9.2      0.
GRID    92      5.2      9.2      0.
GRID    99      1.2      10.     0.
GRID    100     2.      10.     0.
GRID    101     2.8      10.     0.
GRID    102     3.6      10.     0.
GRID    103     4.4      10.     0.
GRID    104     5.2      10.     0.
$

```

Model Definition for Sample Problem (cont.)

```
$
$      se3.blk
$
CQUAD4  14      1      19      20      30      29
CQUAD4  15      1      29      30      36      35
$
GRDSET                                     6
$
$      boundary grids
$
GRID     19      -3.6    4.4    0.
GRID     20      -2.8    4.4    0.
GRID     35      -3.6    6.     0.
GRID     36      -2.8    6.     0.
$
GRID     29      -3.6    5.2    0.
GRID     30      -2.8    5.2    0.
$
```

Model Definition for Sample Problem (cont.)

```

$
$      se4.blk
$
CQUAD4  16      1      27      28
32      31
CQUAD4  17      1      31      32
42      41
$
GRDSET
6
$
$      boundary grids
$
GRID    27      2.8    4.4
0.
GRID    28      3.6    4.4
0.
GRID    41      2.8    6.
0.
GRID    42      3.6    6.
0.
$
GRID    31      2.8    5.2
0.
GRID    32      3.6    5.2
0.
$

```

```

$
$      se5.blk
$
CQUAD4  6      1      9      10      20      19
CQUAD4  7      1      10     11      21      20
CQUAD4  8      1      11     12      22      21
CQUAD4  9      1      12     13      23      22
$
GRDSET
$
$      boundary grids
$
GRID    19      -3.6   4.4    0.
GRID    20      -2.8   4.4    0.
GRID    13      -.4    3.6    0.
GRID    23      -.4    4.4    0.
$
GRID    9       -3.6   3.6    0.
GRID    10      -2.8   3.6    0.
GRID    11      -2.    3.6    0.
GRID    12      -1.2   3.6    0.
GRID    21      -2.    4.4    0.
GRID    22      -1.2   4.4    0.
$

```

Model Definition for Sample Problem (cont.)

```

$
$      se6.blk
$
CQUAD4 10      1      14      15      25      24
CQUAD4 11      1      15      16      26      25
CQUAD4 12      1      16      17      27      26
CQUAD4 13      1      17      18      28      27
$
GRDSET
6
$
$      boundary grids
$
GRID    27      2.8      4.4      0.
GRID    28      3.6      4.4      0.
GRID    14      .4       3.6      0.
GRID    24      .4       4.4      0.
$
GRID    15      1.2      3.6      0.
GRID    16      2.       3.6      0.
GRID    17      2.8      3.6      0.
GRID    18      3.6      3.6      0.
$
GRID    25      1.2      4.4      0.
GRID    26      2.       4.4      0.
$

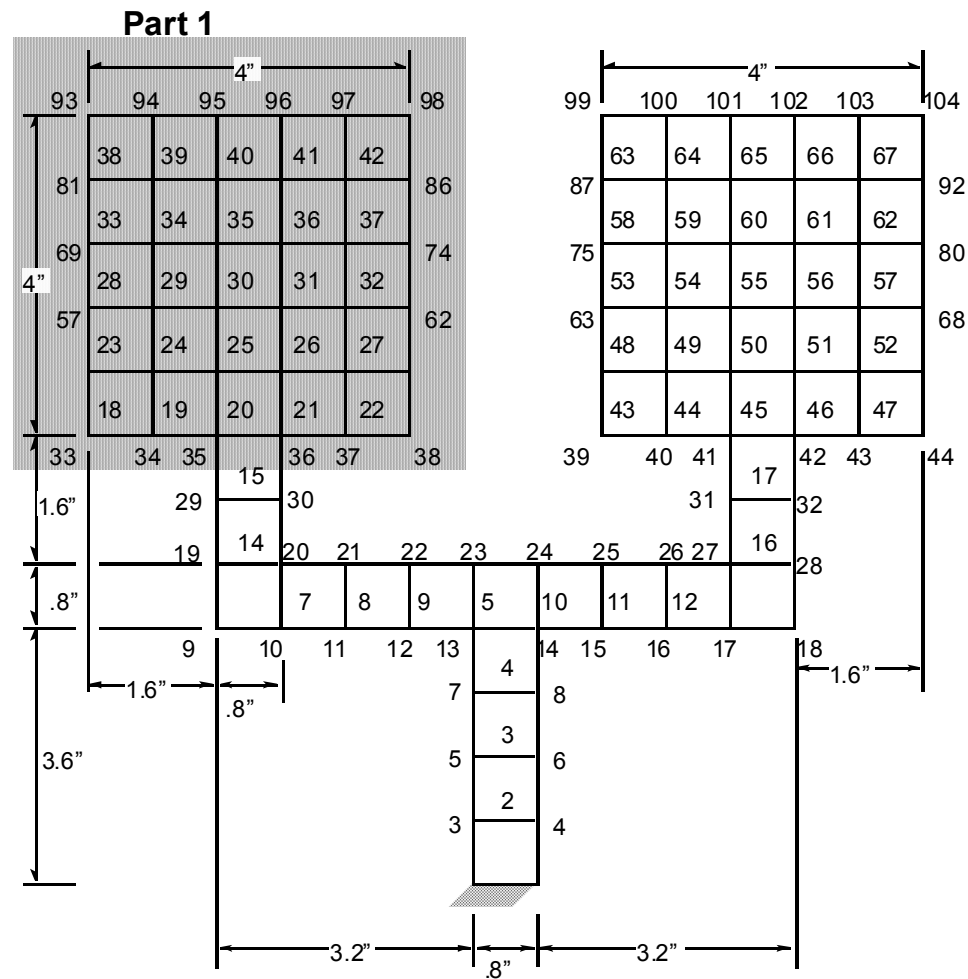
```

```

$
$      se7.blk
$
CQUAD4 1      1      1      2      4      3
CQUAD4 2      1      3      4      6      5
CQUAD4 3      1      5      6      8      7
CQUAD4 4      1      7      8      14     13
$
$
GRID    1      -.4      0.      0.
GRID    2      .4       0.      0.
GRID    3      -.4      0.9     0.
GRID    4      .4       0.9     0.
GRID    5      -.4      1.8     0.
GRID    6      .4       1.8     0.
GRID    7      -.4      2.7     0.
GRID    8      .4       2.7     0.
$
$      boundary grids
$
GRID    13      -.4      3.6     0.
GRID    14      .4       3.6     0.
$
SPC1,1,123456,1,2

```

Sample Problem - Superelement 1



Partitioned Solutions

- **For each superelement, its degrees-of-freedom (DOFs) are divided into two subsets:**
 - Exterior DOFs (called the A-set): Designates the analysis DOFs, which are retained for subsequent processing (for Superelement 1, Grid Points 35 and 36)
 - Interior DOFs: Designates the DOFs that are reduced out during superelement processing and are omitted in subsequent processing (for Superelement 1 of the sample problem, Grid Points 33, 34, 37, 38, 45-50, 57-62, 69-74, 81-86, 93-98).

Partitioned Solutions (cont.)

- **For each superelement, produce a description in matrix terms of its behavior as seen at the boundary or exterior degrees of freedom.**
 - A set of 'G'-sized matrices is produced for each superelement based on the input data.
 - These matrices are reduced down to matrices representing the properties of the superelement as seen by the adjacent (attached) structure.
- **At the residual structure, combine and assemble the boundary matrices.**
 - The BULK DATA for the RESIDUAL structure consists of all 'residual' data not assigned to any superelement plus any common data.
- **Solve for the residual structure displacements.**
- **For each superelement, expand boundary (exterior) displacements to obtain its interior displacements.**

Theory of Static Condensation

After generating matrices and applying MPCs and SPCs,

$$\mathbf{K}_{ff} \mathbf{U}_f = \mathbf{P}_f$$

O-Set = interior points (to be condensed out by the reduction)

A-Set = exterior (or boundary) points (which are retained for further analysis)

Partition

$$\begin{bmatrix} \mathbf{K}_{oo} & \mathbf{K}_{oa} \\ \mathbf{K}_{oa}^T & \mathbf{K}_{aa} \end{bmatrix} \begin{Bmatrix} \mathbf{U}_o \\ \mathbf{U}_a \end{Bmatrix} = \begin{Bmatrix} \mathbf{P}_o \\ \mathbf{P}_a \end{Bmatrix}$$

Theory of Static Condensation (cont.)

- **Extract upper equation and premultiply by k_{oo}^{-1}**

$$k_{oo}^{-1} [k_{oo} U_o + k_{oa} U_a] = k_{oo}^{-1} P_o$$

Let $G_{oa} = -k_{oo}^{-1} k_{oa}$ (Boundary Transformation)

and $U_o^o = k_{oo}^{-1} P_o$ (Fixed Boundary Displacements)

then $U_o = U_o^o + G_{oa} U_a$ (Total Interior Displacements)

Theory of Static Condensation (cont.)

- Substitute expression for U_o in the lower equation

$$K_{oa}^T \left[G_{oa} U_a + U_o^0 \right] + \bar{K}_{aa} U_a = \bar{P}_a$$

then $K_{aa} = K_{oa}^T G_{oa} + \bar{K}_{aa}$ (Boundary Stiffness)

and $P_a = G_{oa}^T P_o + \bar{P}_a$ (Boundary Loads)

- Solve for residual structure

$$U_a = K_{aa}^{-1} P_a \quad (\text{Boundary Displacements})$$

Advantages of Superelement Analysis

- **Large problems (i.e., allows solving problems that exceed your hardware capabilities)**
- **Less CPU or wall clock time per run (reduced risk since each superelement may be processed individually)**
- **Partial redesign requires only partial solution (cost).**
- **Allows more control of resource usage**
- **Partitioned input desirable**
 - Organization
 - Repeated components

Advantages of Superelement Analysis (cont.)

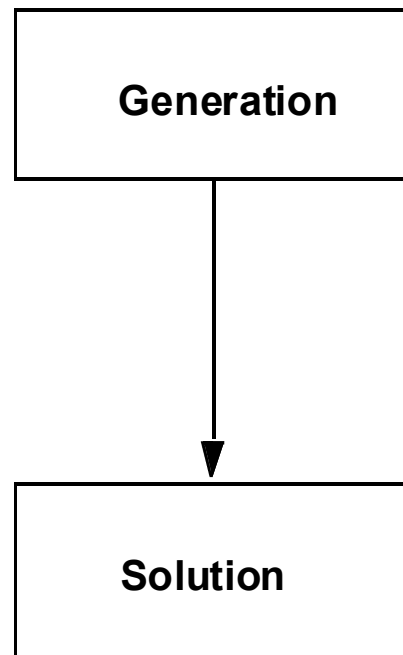
- **Partitioned output desirable**
 - Organization
 - Comprehension
- **Components may be modeled by subcontractors.**
- **Multi-step reduction for dynamic analysis**
- **Zooming (or global-local analysis)**
- **Allows for efficient configuration studies (“What if...”)**

Disadvantages of Superelement Analysis

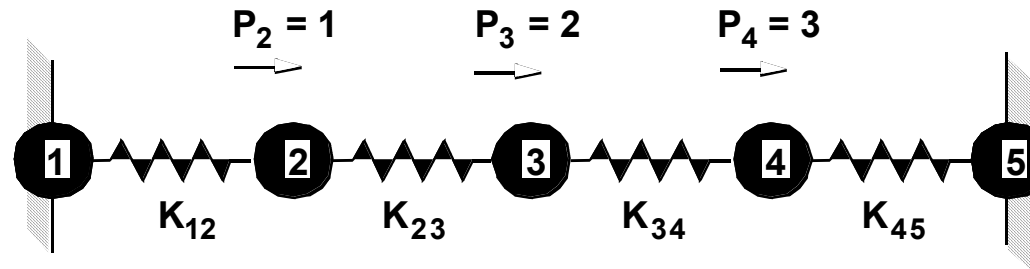
- Increased overhead due to DMAP compilation and database manipulation and storage
- Mandatory static condensation may cancel other cost savings for small models.
- Residual structure is not resequenced and its stiffness matrix is usually dense.
- All superelements must be linear.
- Approximations must be made in dynamics for mass and damping through static, component mode, or generalized dynamic reduction.
- Automatic restarts are available in SOLs 101 and above.

Conventional Analysis

Flowchart



Conventional Analysis (cont.)



K_{xx} = unit stiffness

Generation

$$[K_{GG}] = \begin{bmatrix} K_{12} & -K_{12} & 0 & 0 & 0 \\ -K_{12} & K_{12} + K_{23} & -K_{23} & 0 & 0 \\ 0 & -K_{23} & K_{23} + K_{34} & -K_{34} & 0 \\ 0 & 0 & -K_{34} & K_{34} + K_{45} & -K_{45} \\ 0 & 0 & 0 & -K_{45} & K_{45} \end{bmatrix}$$

$$[K_{GG}] = \begin{bmatrix} 1.0 & -1.0 & 0 & 0 & 0 \\ -1.0 & 2.0 & -1.0 & 0 & 0 \\ 0 & -1.0 & 2.0 & -1.0 & 0 \\ 0 & 0 & -1.0 & 2.0 & -1.0 \\ 0 & 0 & 0 & -1.0 & 1.0 \end{bmatrix}$$

Conventional Analysis (cont.)

Apply Constraints and Solve

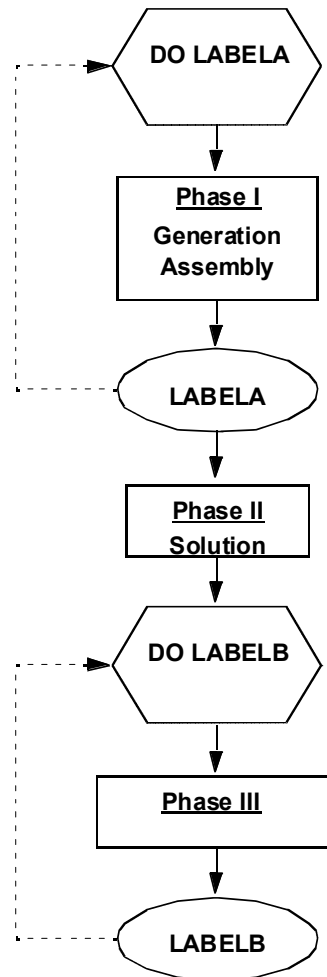
$$\begin{Bmatrix} U_2 \\ U_3 \\ U_4 \end{Bmatrix} = \begin{bmatrix} K_{12} + K_{23} & -K_{23} & \\ -K_{23} & K_{23} + K_{34} & -K_{34} \\ & -K_{34} & K_{34} + K_{45} \end{bmatrix}^{-1} \begin{Bmatrix} P_2 \\ P_3 \\ P_4 \end{Bmatrix}$$

$$\begin{Bmatrix} U_2 \\ U_3 \\ U_4 \end{Bmatrix} = \begin{bmatrix} 2.0 & -1.0 & 0 \\ -1.0 & 2.0 & -1.0 \\ 0 & -1.0 & 2.0 \end{bmatrix}^{-1} \begin{Bmatrix} 1.0 \\ 2.0 \\ 3.0 \end{Bmatrix}$$

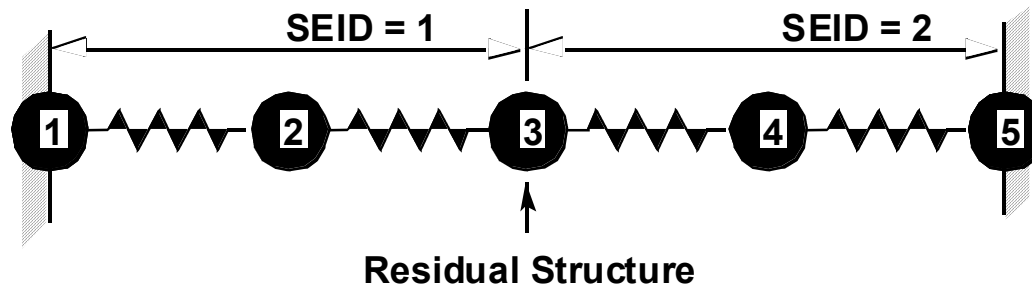
$$\begin{Bmatrix} U_2 \\ U_3 \\ U_4 \end{Bmatrix} = \begin{Bmatrix} 2.5 \\ 4.0 \\ 3.5 \end{Bmatrix}$$

Superelement Analysis

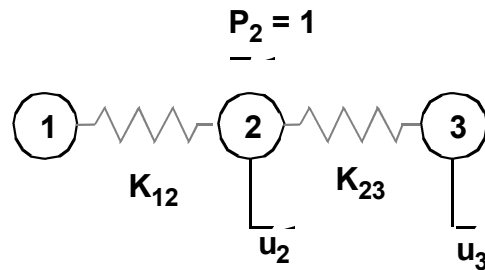
Flowchart



Superelement Analysis (cont.)



Generation - SEID = 1



$$[K_{gg}]^1 = \begin{bmatrix} K_{12} & -K_{12} & 0 \\ -K_{12} & K_{12} + K_{23} & -K_{23} \\ 0 & -K_{23} & K_{23} \end{bmatrix}$$

Superelement Analysis (cont.)

$$\left\{ \mathbf{P}_g \right\}^1 = \left\{ \begin{array}{c} \mathbf{P}_1 \\ \mathbf{P}_2 \\ \bar{\mathbf{P}}_3^1 \end{array} \right\} = \left\{ \begin{array}{c} 0 \\ 1 \\ 0 \end{array} \right\}$$

Reduction - SEID = 1

Eliminate constraints:

$$\begin{aligned} \left[\mathbf{K}_{gg} \right]^1 &= \left[\begin{array}{c|c} \mathbf{K}_{12} + \mathbf{K}_{23} & -\mathbf{K}_{23} \\ \hline -\mathbf{K}_{23} & \mathbf{K}_{23} \end{array} \right] \\ &= \left[\begin{array}{c|c} \mathbf{K}_{oo} & \mathbf{K}_{oa} \\ \hline \mathbf{K}_{ao} & \bar{\mathbf{K}}_{aa} \end{array} \right] \end{aligned}$$

Compute boundary transformation:

$$\begin{aligned} \left[\mathbf{G}_{oa} \right]^1 &= -\left[\mathbf{K}_{oo} \right]^{-1} \left[\mathbf{K}_{oa} \right] \\ &= \frac{\mathbf{K}_{23}}{\mathbf{K}_{12} + \mathbf{K}_{23}} = 0.5 \end{aligned}$$

Superelement Analysis (cont.)

Compute boundary stiffness:

$$[K_{aa}]^1 = [K_{aa} + K_{oa}^T G_{oa}]$$

$$K_{aa}^1 = \frac{K_{12}K_{23}}{K_{12} + K_{23}} = 0.5$$

Compute boundary loading:

$$\begin{aligned} \left\{ P_f \right\}^1 &= \left\{ \begin{matrix} P_2 \\ \bar{P}_3 \end{matrix} \right\} = \left\{ \begin{matrix} 1 \\ 0 \end{matrix} \right\} \\ &= \left\{ \begin{matrix} P_o \\ \bar{P}_a \end{matrix} \right\} \\ \left\{ P_a \right\}^1 &= \left\{ \bar{P}_a + G_{oa}^T P_o \right\} \\ P_3^1 &= \cancel{\bar{P}_3}^0 + \frac{K_{23}}{K_{12} + K_{23}} P_2 = 0.5 \end{aligned}$$

Superelement Analysis (cont.)

Similarly - SEID = 2

$$[K_{gg}]^2 = \begin{bmatrix} K_{34} & -K_{34} & 0 \\ -K_{34} & K_{34} + K_{45} & -K_{45} \\ 0 & -K_{45} & K_{45} \end{bmatrix}$$

$$\left\{ \begin{matrix} P_g \end{matrix} \right\}^2 = \left\{ \begin{matrix} \bar{P}_3^2 \\ P_4 \\ P_5 \end{matrix} \right\} = \left\{ \begin{matrix} 0 \\ 3 \\ 0 \end{matrix} \right\}$$

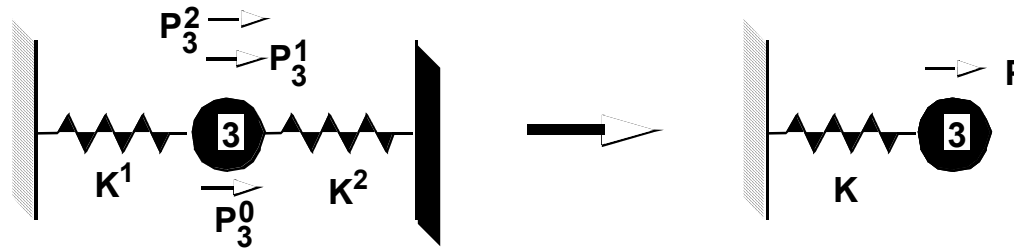
$$[G_{oa}]^2 = \frac{K_{34}}{K_{34} + K_{45}} = 0.5$$

$$[K_{aa}]^2 = \frac{K_{34} K_{45}}{K_{34} + K_{45}} = 0.5$$

$$P_3^2 = \cancel{\bar{P}_3^2}^0 + \frac{K_{34}}{K_{34} + K_{45}} P_2 = 1.5$$

Superelement Analysis (cont.)

Residual Structure



Assembly:

$$[K_{aa}] = [K_{aa}^1 + K_{aa}^2 + K_{gg}^0]$$

$$K = K^1 + K^2 = 1$$

$$\{P_a\} = \{P_a^1 + P_a^2 + P_g^0\}$$

$$P = P_3^1 + P_3^2 + P_3^0 = 4$$

Solution:

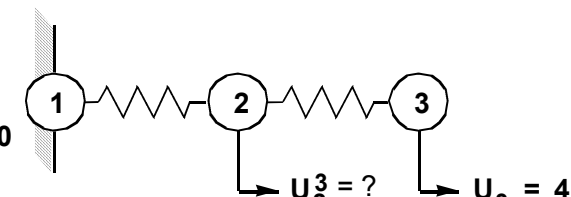
$$\{U_a\} = [K_{aa}]^{-1} \{P_a\}$$

$$U_3^0 = \frac{P}{K} = 4$$

Superelement Analysis (cont.)

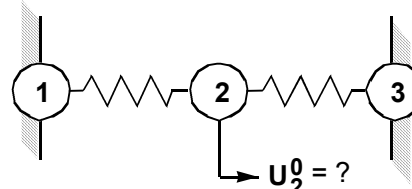
- Data Recovery - SEID = 1
- Enforce (transform) boundary motion.

$$\begin{Bmatrix} U_o^a \end{Bmatrix} = [G_{oa}] \begin{Bmatrix} U_a \end{Bmatrix}$$

$$U_2^3 = \frac{K_{23}}{K_{12} + K_{23}} U_3 = 2.0$$


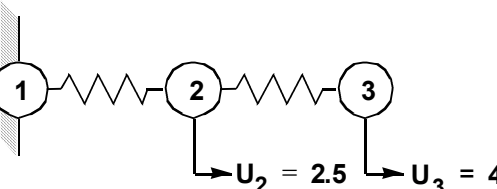
- Compute fixed-boundary motion.

$$\begin{Bmatrix} U_o^0 \end{Bmatrix} = [K_{oo}]^{-1} \begin{Bmatrix} P_o \end{Bmatrix}$$

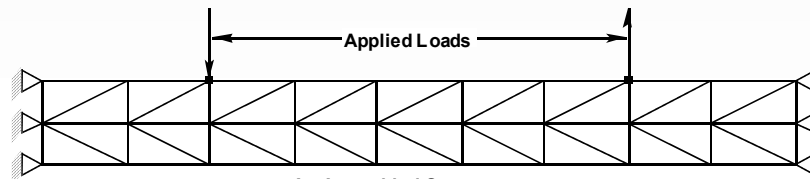
$$U_2^0 = \frac{1}{K_{12} + K_{23}} P_2 = 0.5$$


- Compute total motion.

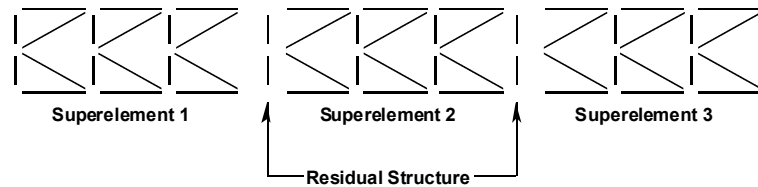
$$\begin{Bmatrix} U_o \end{Bmatrix} = \begin{Bmatrix} U_o^0 \end{Bmatrix} + \begin{Bmatrix} U_o^a \end{Bmatrix}$$

$$U_2 = \frac{K_{23} \times U_3 + P_2}{K_{12} + K_{23}} = 2.5$$


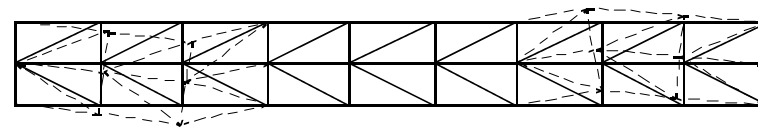
Superposition of Partial Solutions



A. Assembled Structure

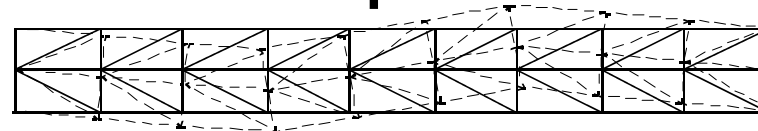


B. Partitioned Structure



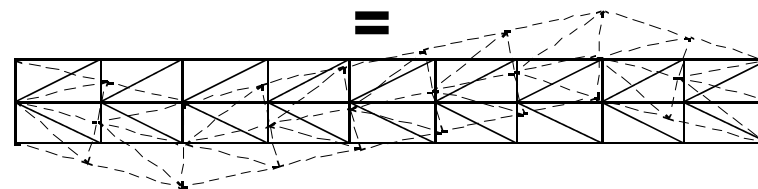
C. Partial Solution Due to External Loads

+



D. Partial Solution Due to Boundary Motion

=



E. Assembled Solution

Superelement Reduction Methods Available in Dynamic Analysis

- **Static reduction**
 - Static condensation of stiffness and Guyan reduction of mass
 - Automatically done
- **Dynamic reduction**
 - Component modal synthesis (CMS)
 - Analytical (SOL 103)

Degrees of Reduction

- **Static reduction (default)**
 - Interior masses relumped to boundary (Guyan)
 - Rigid body properties preserved
 - Important masses must be made exterior (boundary)
- **Component mode reduction - in addition to static reduction**
 - Interior masses represented by exact eigenvectors of the component
 - Eigensolutions for each superelement may be output

Guyan or Static Reduction

- Based on stiffness
- Perform static condensation on stiffness

$$\begin{bmatrix} \mathbf{K}_{oo} & \mathbf{K}_{oa} \\ \mathbf{K}_{oa}^T & \mathbf{K}_{aa} \end{bmatrix} \begin{Bmatrix} \mathbf{U}_o \\ \mathbf{U}_a \end{Bmatrix} = \begin{Bmatrix} \mathbf{P}_o \\ \mathbf{P}_a \end{Bmatrix}$$

If $\mathbf{P}_o=0$, then $\{\mathbf{u}_o\} = [\mathbf{G}_{oa}] \{\mathbf{u}_a\}$, where $\mathbf{G}_{oa} = -[\mathbf{K}_{oo}^{-1}] [\mathbf{K}_{oa}]$

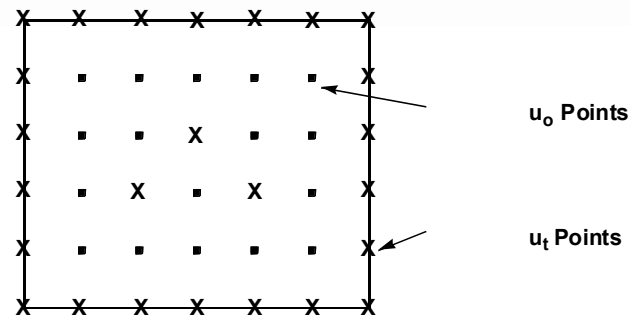
Use this transformation to go from the F-set to the A-set

$$\begin{aligned} [\mathbf{K}_{aa}] &= [\bar{\mathbf{K}}_{aa}] + [\mathbf{K}_{oa}]^T [\mathbf{G}_{oa}] \\ [\mathbf{M}_{aa}] &= [\bar{\mathbf{M}}_{aa}] + [\mathbf{M}_{oa}]^T [\mathbf{G}_{oa}] + [\mathbf{G}_{oa}]^T [\mathbf{M}_{oa}] + [\mathbf{G}_{oa}]^T [\mathbf{M}_{oo}] [\mathbf{G}_{oa}] \end{aligned}$$

However, $\{\mathbf{U}_o\}$ (internal dynamic effects) is ignored

- No approximation if no masses, damping, or applied loads are specified on O-set (interior DOFs)
- Good approximation if component frequencies are much higher than residual structure frequencies and the excitation frequencies

Comparison of Reduction Methods



- **Static reduction**

$$\begin{Bmatrix} U_o \end{Bmatrix} = [G_{ot}] \begin{Bmatrix} U_t \end{Bmatrix} + \begin{Bmatrix} \cancel{U_o^0} \end{Bmatrix}$$

Local dynamic effects are ignored.

- **Generalized dynamic reduction**

$$\begin{Bmatrix} U_o \end{Bmatrix} = \begin{bmatrix} G_{ot} & G_{oq} \end{bmatrix} \begin{Bmatrix} U_t \\ \dot{U}_q \end{Bmatrix}$$

Approximate eigenvectors are used to represent the interior motion.

- **Component mode reduction**

$$\begin{Bmatrix} U_o \end{Bmatrix} = \begin{bmatrix} G_{ot} & G_{oq} \end{bmatrix} \begin{Bmatrix} U_t \\ \dot{U}_q \end{Bmatrix}$$

- **Exact eigenvectors are used to represent the interior motion.**

Advantages of each Reduction Method

- **Advantages of Component Mode Reduction over Static Reduction**
 - Can use experimental results
 - More accurate for the same number of dynamic DOFs
 - Ideal for highly coupled and uncoupled structures
- **Advantages of Static Reduction over Component Mode Reduction**
 - Cheaper
 - Less sophisticated
 - Fewer problems

Calculation of Normal Modes using Static Reduction Only

- **No generalized coordinates are needed for the superelements. (exception: residual structure, if component reduction at the residual structure is desired)**
- **Superelement mass, damping, and stiffness are reduced statically to exterior DOFs.**
- **ASETi and QSETi entries may be specified for residual structure DOFs only.**
- **If no ASETi entries are present, then all DOFs interior to the residual structure are retained for the eigensolution.**
- **If ASETi entries are present, then only those DOFs specified on the ASETi entries are retained for the eigensolution.**
- **Case Control is similar to static analysis with the addition of a METHOD command under the residual structure subcase.**

Calculation of Normal Modes using Dynamic Reduction

- The behavior of a superelement is represented by its component modes in addition to the static shapes.
- The superelement stiffness, mass, and damping are transformed using both physical and modal variables.
- The superelement modes are computed if a METHOD command appears under the superelement subcase.
- The number of superelement modes computed (modal truncation) is controlled by the EIGR or EIGRL entry.
- The number of superelement modes sent downstream directly to the residual is done one of three ways:
 - Param,autoqset,yes
 - SENQSET DOFs
 - QSETi and SPOINT DOFs.
- If the superelement modes are to be sent downstream to another superelement, QSETi and SPOINTs must be used. These SPOINTs must also be “connected” to the SPOINTs in the downstream superelement using the SECONCT entry.

Calculation of Normal Modes using Dynamic Reduction (cont.)

- By default, superelement modes are computed with all exterior degrees of freedom fixed (in the B-set). This is better known as the Craig-Bampton method.
- Superelement modes are computed in Phase I under the SEMR operation.
- Copied superelements need to have the same number of exterior DOFs as their primary. If the primary has an SENQSET, then the image must have equivalent DOFs to represent the modes.

Superelement Internal Generalized Degree of Freedom

SENQSET

Defines number of internally generated scalar points for superelement dynamic reduction.

Format:

1	2	3	4	5	6	7	8	9	10
SENQSET	SEID	N							

Example:

SENQSET	110	45							

Field	Contents
SEID	Superelement identification number. See Remark 3.. (Integer>0 or Character="ALL")
N	Number of internally generated scalar points for dynamic reduction generalized coordinates. (Integer>0; Default=0)

Remarks:

1. SENQSET can only be specified in the main Bulk Data Section and is ignored after the BEGIN SUPER=n command.
2. SENQSET is only required if the user wants to internally generated scalar points used for dynamic reduction.
3. SEID="ALL" will automatically generate N q-set degrees of freedom for all superelements, except the residual structure (SEID=0). Specifying additional SENQSET entries for specific superelements will override the value of N specified on this entry.
4. If the user manually specifies q-set degrees of freedom using a SEQSETi or QSETi entries, then the internally generated scalar points will not be generated.LQ

Bulk Data Entry

Scalar Point Definition

SPOINT

Defines scalar points.

Format:

1	2	3	4	5	6	7	8	9	10
SPOINT	ID1	ID2	ID3	ID4	ID5	ID6	ID7	ID8	

Example:

SPOINT	3	18	1	4	16	2			

Alternate Format and Example:

SPOINT	ID1	"THRU"	ID2						
SPOINT	5	THRU	649						

Field

Contents

IDi Scalar point identification number. (0<Integer<1000000; For "THRU" option, ID1<ID2)

Remarks:

1. A scalar point defined by its appearance on the connection entry for a scalar element (see the CELASi, CMASSi, and CDAMPi entries) need not appear on an SPOINT entry.
2. All scalar point identification numbers must be unique with respect to all other structural, scalar, and fluid points. However, duplicate scalar point identification numbers are allowed in the input.
3. This entry is used primarily to define scalar points appearing in single-point or multipoint constraint equations to which no scalar elements are connected.
4. If the alternate format is used, all scalar points ID1 through ID2 are defined.
5. For a discussion of scalar points, see the *MSC.Nastran Reference Manual*, Section 5.6.

Bulk Data Entry

Generalized Degree of Freedom

QSET

Defines generalized degrees of freedom (q-set) to be used for dynamic reduction or component mode synthesis

Format:

1	2	3	4	5	6	7	8	9	10
QSET	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

Example:

QSET	15	123456	1	7	9	2	105	6	

Field

Contents

IDi

Grid or scalar point identification number. (Integer>0)

Ci

Component number. (Integer zero or blank for scalar points or any unique combination of the Integers 1 through 6 for grid points with no embedded blanks.)

Remarks:

1. Degrees of freedom specified on this entry form members of the mutually exclusive q-set. They may not be specified on other entries that define mutually exclusive sets. See the MSC.Nastran Quick Reference Guide, Appendix B for a list of these entries.
2. Degrees of freedom specified on QSET and QSET1 entries are automatically placed in the a-set.
3. When ASET, ASET1, QSET, and/or QSET1 entries are present, all degrees of freedom not otherwise constrained (e.g., SPCi or MPC entries) will be placed in the omitted set (o-set).

Bulk Data Entry

Generalized Degree of Freedom (Alternate Form of QSET Entry)

QSET1

Defines generalized degrees of freedom (q-set) to be used for generalized dynamic reduction or component mode synthesis.

Format:

	1	2	3	4	5	6	7	8	9	10
QSET1	C	ID1	ID2	ID3	ID4	ID5	ID6	ID7		
		ID8	ID9	-etc.-						

Example:

QSET1	123456	1	7	9	22	105	6	22		
	52	53								

Alternate Format and Example:

QSET1	C	ID1	"THRU"	ID2						
QSET1	0	101	THRU	110						

Field

Contents

C

Component number. (Integer zero or blank for scalar points or any unique combination of the Integers 1 through 6 for grid points with no embedded blanks.)

IDi

Grid or scalar point identification number. (Integer>0; For THRU option, ID1<ID2.)

Remarks:

1. Degrees of freedom specified on this entry form members of the mutually exclusive q-set. They may not be specified on other entries that define mutually exclusive sets. See the MSC.Nastran Quick Reference Guide, Appendix B for a list of these entries.
2. Degrees of freedom specified on QSET and QSET1 entries are automatically placed in the a-set.
3. When ASET, ASET1, QSET, and/or QSET1 entries are present, all degrees of freedom not otherwise constrained (e.g., SPCi or MPC entries) will be placed in the omitted set (o-set).

Bulk Data Entry

Calculation of Normal Modes using Dynamic Reduction (cont.)

- **For free-free superelement component modes, all exterior DOFs should be specified on CSETi entries.**
 - The rigid-body modes ($f=0.0$ Hz) are a linear combination of the static vectors and should not be included in the reduction.
 - Either:
 - Do not calculate them ($F1>0.0$ on the EIGR or EIGRL entry).
 - Calculate them and hope that the program will remove them (see PARAM,EPSRC in the *MSC Nastran Reference Manual*).
- **Mixed-boundary modes may be calculated by using the CSETi and BSETi entries to describe the exterior DOFs to be unconstrained and constrained during CMS.**
 - If 0.0 Hz mixed boundary modes exist, they must be handled in a similar manner to those in the free-free case.

Fixed Analysis Degrees of Freedom

BSET

Defines analysis set (a-set) degrees of freedom to be fixed (b-set) during generalized dynamic reduction or component mode synthesis calculations.

Format:

1	2	3	4	5	6	7	8	9	10
BSET	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

Example:

BSET	2	135	14	6					

Field	Contents
IDi	Grid or scalar point identification number. (Integer>0)
Ci	Component number. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points. No embedded blanks.)

Remarks:

1. If there are no CSETi or BSETi entries present, all a-set points are considered fixed during component mode analysis. If there are only BSETi entries present, any a-set degrees of freedom not listed are placed in the free boundary set (c-set). If there
2. Degrees of freedom specified on this entry form members of the mutually exclusive b-set. They may not be specified on other entries that define mutually exclusive sets. See the MSC.Nastran Quick Reference Guide, Appendix B for a list of these entries.
3. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees of freedom will be reassigned as follows:
 - a. If there are no o-set (omitted) degrees of freedom, then singular b-set and c-set degrees of freedom are reassigned to the s-set.
 - b. If there are o-set (omitted) degrees of freedom, then singular c-set degrees of freedom are reassigned to the b-set. Singular b-set degrees of freedom are not reassigned.

Bulk Data Entry

Fixed Analysis Degrees of Freedom, Alternate Form of BEST Entry**BSET1**

Defines analysis set (a-set) degrees of freedom to be fixed (b-set) during generalized dynamic reduction or component mode synthesis calculations.

Format:

1	2	3	4	5	6	7	8	9	10
BSET1	C	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	ID9	ID10	-etc.-					

Example:

BSET1	2	135	14	6	23	24	25	26	
	122	127							

Alternate Format and Example:

BSET1	C	ID1	"THRU"	ID2					
BSET1	3	6	THRU	32					

Field**Contents**

C

Component numbers. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points with no embedded blanks.)

IDi

Grid or scalar point identification numbers. (Integer>0; For "THRU" option, ID1<ID2)

Remarks:

1. If there are no CSETi or BSETi entries present, all a-set points are considered fixed during component mode analysis. If there are only BSETi entries present, any a-set degrees of freedom not listed are placed in the free boundary set (c-set). If there
2. Degrees of freedom specified on this entry form members of the mutually exclusive b-set. They may not be specified on other entries that define mutually exclusive sets. See the MSC.Nastran Quick Reference Guide, Appendix B for a list of these entries.
3. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees of freedom will be reassigned as follows:
 - a. If there are no o-set (omitted) degrees of freedom, then singular b-set and c-set degrees of freedom are reassigned to the s-set.
 - b. If there are o-set (omitted) degrees of freedom, then singular c-set degrees of freedom are reassigned to the b-set. Singular b-set degrees of freedom are not reassigned.

Bulk Data Entry

Free Boundry Degree of Freedom

CSET

Defines analysis set (a-set) degrees of freedom to be free (c-set) during generalized dynamic reduction or component modes calculations.

Format:

1	2	3	4	5	6	7	8	9	10
CSET	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

Example:

CSET	124	1	5	23	6	16			

Field

Contents

Idi	Grid or scalar point identification number. (Integer>0)
Ci	Component numbers. (Integer zero or blank for scalar points, or any unique combination of the Integers 1 through 6 for grid points with no embedded blanks.)

Remarks:

1. If there are no CSETi or BSETi entries present, all a-set degrees of freedom are considered fixed during component modes analysis. If there are only BSETi entries present, any a-set degrees of freedom not listed are placed in the free boundary set (c-set)
2. Degrees of freedom specified on this entry form members of the mutually exclusive c-set. They may not be specified on other entries that define mutually exclusive sets. See the MSC.Nastran Quick Reference Guide, Appendix B for a list of these entries.
3. If PARAM,AUTOSPC is YES then singular b-set and c-set degrees of freedom will be reassigned as follows:
 - a. If there are no o-set (omitted) degrees of freedom, then singular b-set and c-set degrees of freedom are reassigned to the s-set.
 - b. If there are o-set (omitted) degrees of freedom, then singular c-set degrees of freedom are reassigned to the b-set. Singular b-set degrees of freedom are not reassigned.

Bulk Data Entry

Free Boundry Degree of Freedom, Alternate Form of CSET Entry

CSET1

Defines analysis set (a-set) degrees of freedom to be free (c-set) during generalized dynamic reduction or component modes calculations.

Format:

1	2	3	4	5	6	7	8	9	10
CSET1	C	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	ID9	-etc.-						

Example:

CSET1	124	1	5	7	6	9	12	122	
	127								

Alternate Formats and Examples:

CSET1	C	ID1	"THRU"	ID2					
CSET1	3	6	THRU	32					

CSET1		"ALL"							
CSET1		ALL							

Field	Contents
C	Component number. (Integer zero or blank for scalar points, or any unique combination of the Integers 1 through 6 for grid points with no embedded blanks.)
IDi	Grid or scalar point identification number. (Integer>0; For THRU option, ID1<ID2)

Remarks:

1. If there are no CSETi or BSETi entries present, all a-set degrees of freedom are considered fixed during component modes analysis. If there are only BSETi entries present, any a-set degrees of freedom not listed are placed in the free boundary set (c-set)

(Continued)

Bulk Data Entry

2. Degrees of freedom specified on this entry form members of the mutually exclusive c-set. They may not be specified on other entries that define mutually exclusive sets. See the MSC.Nastran Quick Reference Guide, Appendix B for a list of these entries.
3. If PARAM,AUTOSPC is YES then singular b-set and c-set degrees of freedom will be reassigned as follows:
 - a. If there are no o-set (omitted) degrees of freedom, then singular b-set and c-set degrees of freedom are reassigned to the s-set.
 - b. If there are o-set (omitted) degrees of freedom, then singular c-set degrees of freedom are reassigned to the b-set. Singular b-set degrees of freedom are not reassigned.

Default CMS Method “Fixed Boundary” CMS

Description of Methodology (better known as Craig-Bampton CMS)

- The superelement matrices are partitioned into two sets of degrees of freedom (DOFs). The first set (the B-set) represents the boundary points. The second set is the interior DOFs (the O-set).
- A set of “constraint” modes is generated. Each “constraint” mode represents the motion of the model resulting from moving one boundary DOF 1.0 unit, while holding the other boundary DOF fixed. Therefore, there is one “constraint” mode for each boundary DOF (these vectors are known as GOAT in MSC Nastran)

- In matrix form,
$$\begin{bmatrix} \mathbf{K}_{oo} & \mathbf{K}_{ob} \\ \mathbf{K}_{bo} & \mathbf{K}_{bb} \end{bmatrix} \begin{Bmatrix} \phi_{ob} \\ \bar{\mathbf{I}}_{bb} \end{Bmatrix} = \begin{Bmatrix} \mathbf{0} \\ \bar{\mathbf{P}}_b \end{Bmatrix}$$

($\mathbf{P}_b$ is not actually applied.)

- The first line gives
$$\begin{Bmatrix} \phi_{ob} \end{Bmatrix} = -[\mathbf{K}_{oo}]^{-1} [\mathbf{K}_{ob}] \begin{Bmatrix} \bar{\mathbf{I}}_{bb} \end{Bmatrix} \quad (\mathbf{G}_{OAT})$$

Default CMS Method “Fixed Boundary” CMS (cont.)

- giving the following “constraint” modes:

$$\left\{ \phi_b \right\} = \left\{ \begin{array}{c} \phi_{ob} \\ \hline \mathbf{I}_{bb} \end{array} \right\}$$

- Now the O-set equations are solved for the “fixed-boundary” modes $\{\phi_{oo}\}$ (known as GOAQ in MSC Nastran).

$$-\omega_k^2 [\mathbf{M}_{oo}] \left\{ \phi_{oo} \right\} + [\mathbf{K}_{oo}] \left\{ \phi_{oo} \right\} = \mathbf{0}$$

As many fixed-boundary modes as are desired are found. Then they are concatenated with the “constraint” modes to form the generalized coordinates.

$$\left\{ \phi_G \right\} = \left\{ \begin{array}{c|c} \phi_{ob} & \phi_{oo} \\ \hline \mathbf{I}_{bb} & \mathbf{0} \end{array} \right\}$$

- The mass and stiffness matrices are pre- and post multiplied by these modes to obtain the “generalized” mass and stiffness

$$[\mathbf{K}_G] = \{\phi_G\}^T [\mathbf{K}_{ff}] \{\phi_G\}$$

$$[\mathbf{M}_G] = \{\phi_G\}^T [\mathbf{M}_{ff}] \{\phi_G\}$$

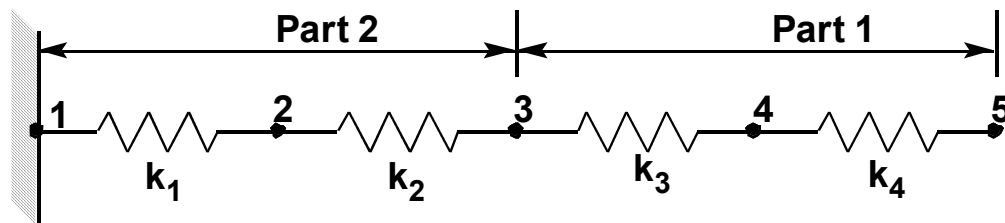
where the F-set is the union of the B- and O-sets.

Default CMS Method “Fixed Boundary” CMS (cont.)

- These “generalized” matrices contain physical DOFs representing the boundaries and “modal” coordinates representing the “fixed-boundary” component modes.
- At this point, these matrices can be treated like any other structural matrices, and data recovery can be performed for the component in a manner similar to using modal coordinates. That is, the displacements of the generalized coordinates are multiplied by the associated vectors and added together to obtain the component displacements.
- The calculated modes for each superelement are internally scaled to have a maximum displacement = 1.0 in MSC Nastran (regardless of the scaling requested by the user).

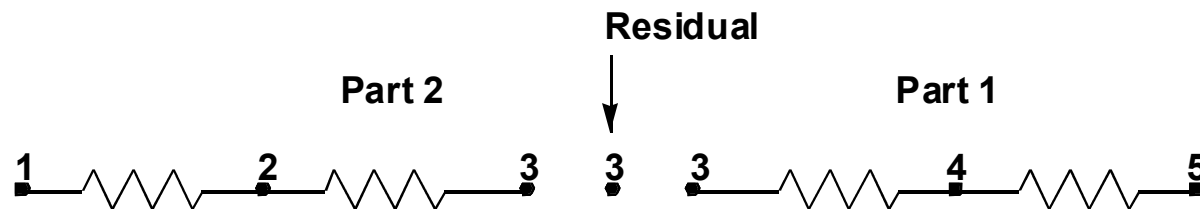
Solution by Hand

Component Modal Synthesis Sample



$$k_1 = k_2 = k_3 = k_4 = 1.0$$

$$m_1 = m_2 = m_4 = m_5 = 1.0 ; \quad m_3 = 1.0$$



Theoretical solution for frequencies

i	1	2	3	4
f_i	0.0553	0.1592	0.2438	0.2991
$\lambda_i = \omega^2$	0.1206	1	2.3473	3.5321

Solution by Hand (cont.)

- Superelement 1

$$K_{gg} = \begin{bmatrix} 1.0 & -1.0 & 0 \\ -1.0 & 2.0 & -1.0 \\ 0 & -1.0 & 1.0 \end{bmatrix} \quad M_{gg} = \begin{bmatrix} 0 & 0 & 0 \\ 0 & 1.0 & 0 \\ 0 & 0 & 1.0 \end{bmatrix} \left\{ \begin{array}{c} U_3 \\ U_4 \\ U_5 \end{array} \right\}$$

- Grid Point 3 is the “boundary” point; solve for “constraint” modes.

$$\text{Where } K_{oo} = \begin{bmatrix} 1.0 & -1.0 & 0 \\ -1.0 & 2.0 & -1.0 \\ 0 & -1.0 & 1.0 \end{bmatrix} \left\{ \begin{array}{c} 1.0 \\ U_4 \\ U_5 \end{array} \right\} = \left\{ \begin{array}{c} P_b \\ 0 \\ 0 \end{array} \right\}$$

$$K_{ob} = \begin{bmatrix} 2.0 & -1.0 \\ -1.0 & 1.0 \end{bmatrix}$$

$$= \begin{bmatrix} -1.0 \\ 0 \end{bmatrix}$$

$$K_{oo}^{-1} = \begin{bmatrix} 1.0 & 1.0 \\ 1.0 & 2.0 \end{bmatrix}$$

Solution by Hand (cont.)

- **Where**

$$\phi_{ob} = -\begin{pmatrix} 1.0 & 1.0 \\ 1.0 & 2.0 \end{pmatrix} \begin{Bmatrix} 1.0 \\ 0.0 \end{Bmatrix} = \begin{Bmatrix} 1.0 \\ 1.0 \end{Bmatrix}$$

$$\phi_b = \begin{Bmatrix} 1.0 \\ 1.0 \\ 1.0 \end{Bmatrix}$$

- **Solve for “fixed-boundary” modes.**
- **Note: Internally MSC Nastran uses component modes scaled to a maximum deformation of 1.0. Output for the component modes is based on the normalization performed by the eigenvalue solution.**

$$\left[-\omega^2 \mathbf{M}_{oo} + \mathbf{K}_{oo} \right] \begin{bmatrix} \phi_{oo} \end{bmatrix} = \mathbf{0} = \left\{ \begin{bmatrix} -\omega^2 & 0 \\ 0 & -\omega^2 \end{bmatrix} + \begin{bmatrix} 2.0 & -1.0 \\ -1.0 & 1.0 \end{bmatrix} \right\} \begin{bmatrix} \phi_{oo} \end{bmatrix}$$

Solution by Hand (cont.)

$$\det \begin{bmatrix} 2 - \omega^2 & -1 \\ -1 & 1 - \omega^2 \end{bmatrix} = 0 \longrightarrow \begin{aligned} \omega^2 &= 0.3819, 2.618 \\ f &= 0.098 \text{ Hz}, 0.2575 \text{ Hz} \end{aligned}$$

$$\phi_1 = \begin{Bmatrix} 0.6180 \\ 1.0000 \end{Bmatrix} \quad \phi_1 = \begin{Bmatrix} 0.5257 \\ 0.8506 \end{Bmatrix} \quad \left. \vphantom{\begin{Bmatrix} 0.5257 \\ 0.8506 \end{Bmatrix}} \right\} \text{Normalized to unit mass}$$

$$\phi_2 = \begin{Bmatrix} -1.0000 \\ 0.6180 \end{Bmatrix} \quad \phi_2 = \begin{Bmatrix} -0.8506 \\ 0.5257 \end{Bmatrix}$$

$$\phi_{00} = \begin{bmatrix} 0.6180 & -1.0000 \\ 1.0000 & 0.6180 \end{bmatrix}$$

$$\phi_G = \begin{bmatrix} 1.000 & 0 & 0 \\ 1.000 & 0.618 & -1.000 \\ 1.000 & 1.000 & 0.618 \end{bmatrix}$$

$$\begin{bmatrix} \phi_G \end{bmatrix}^T \begin{bmatrix} K_{gg} \end{bmatrix} \begin{bmatrix} \phi_G \end{bmatrix} = \begin{bmatrix} 0 & 0 & 0 \\ 0 & 0.5279 & 0 \\ 0 & 0 & 3.6180 \end{bmatrix} \begin{matrix} u_3 \\ u_{1001} \\ u_{1002} \end{matrix}$$

$$\begin{bmatrix} \phi_G \end{bmatrix}^T \begin{bmatrix} M_{gg} \end{bmatrix} \begin{bmatrix} \phi_G \end{bmatrix} = \begin{bmatrix} 2.0000 & 1.6180 & -0.3820 \\ 1.6180 & 1.3820 & 0 \\ -0.3820 & 0 & 1.3820 \end{bmatrix} \begin{matrix} u_3 \\ u_{1001} \\ u_{1002} \end{matrix}$$

where 1001 and 1002 are scalar points used to represent Superelement 1's modes.

Solution by Hand (cont.)

Superelement 2

$$\mathbf{K}_{gg} = \begin{bmatrix} 1.0 & -1.0 & 0 \\ -1.0 & 2.0 & -1.0 \\ 0 & -1.0 & 1.0 \end{bmatrix} \quad \mathbf{M}_{gg} = \begin{bmatrix} 1.0 & 0 & 0 \\ 0 & 1.0 & 0 \\ 0 & 0 & 0 \end{bmatrix}$$

- Apply constraint to grid point 1.

$$\phi_b = \begin{Bmatrix} 0.5 \\ 1.0 \end{Bmatrix}$$

$$\{\phi_{oo}\} = \{1\} \quad \omega^2 = 2.0 \quad \mathbf{f} = 0.2251 \quad \begin{bmatrix} \phi_G \end{bmatrix} = \begin{bmatrix} 0.5 & 1.0 \\ 1.0 & 0 \end{bmatrix}$$

$$\begin{bmatrix} \phi_G \end{bmatrix}^T \mathbf{K}_{gg} \begin{bmatrix} \phi_G \end{bmatrix} = \begin{bmatrix} 0.5 & 0 \\ 0 & 2.0 \end{bmatrix} \begin{matrix} u_3 \\ u_{1005} \end{matrix}$$

$$\begin{bmatrix} \phi_G \end{bmatrix}^T \mathbf{M}_{gg} \begin{bmatrix} \phi_G \end{bmatrix} = \begin{bmatrix} 0.25 & 0.50 \\ 0.50 & 1.00 \end{bmatrix} \begin{matrix} u_3 \\ u_{1005} \end{matrix}$$

where 1005 is a scalar point used to represent Superelement 2's mode.

Solution by Hand (cont.)

Residual Structure

- Before adding superelements:

$$\mathbf{K}_{gg} = \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}$$

$$\mathbf{M}_{gg} = \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{matrix} u_3 \\ u_{1001} \\ u_{1002} \\ u_{1005} \end{matrix}$$

Solution by Hand (cont.)

- Add Superelement 1

$$\mathbf{K}_{gg} = \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 0.5279 & 0 & 0 \\ 0 & 0 & 3.618 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}$$

$$\mathbf{M}_{gg} = \begin{bmatrix} 3.0000 & 1.6180 & -0.3820 & 0 \\ 1.6180 & 1.3820 & 0 & 0 \\ -0.3820 & 0 & 1.3820 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{matrix} u_3 \\ u_{1001} \\ u_{1002} \\ u_{1005} \end{matrix}$$

Solution by Hand (cont.)

- Add Superelement 2

$$\mathbf{K}_{gg} = \begin{bmatrix} 0.5 & 0 & 0 & 0 \\ 0 & 0.5279 & 0 & 0 \\ 0 & 0 & 3.618 & 0 \\ 0 & 0 & 0 & 2.0 \end{bmatrix}$$

$$\mathbf{M}_{gg} = \begin{bmatrix} 3.2500 & 1.6180 & -0.3820 & 0.5 \\ 1.6180 & 1.3820 & 0 & 0 \\ -0.3820 & 0 & 1.3820 & 0 \\ 0 & 0 & 0 & 1.0 \end{bmatrix} \begin{matrix} u_3 \\ u_{1001} \\ u_{1002} \\ u_{1005} \end{matrix}$$

Solution by Hand (cont.)

$$\mathbf{K}_{ff} = \begin{bmatrix} 0.5 & 0 & 0 & 0 \\ 0 & 0.5279 & 0 & 0 \\ 0 & 0 & 3.618 & 0 \\ 0 & 0 & 0 & 2.0 \end{bmatrix}$$

$$\mathbf{M}_{ff} = \begin{bmatrix} 3.2500 & 1.6180 & -0.3820 & 0 \\ 1.6180 & 1.3820 & 0 & 0 \\ -0.3820 & 0 & 1.3820 & 0 \\ 0 & 0 & 0 & 1.0 \end{bmatrix} \begin{matrix} u_3 \\ u_{1001} \\ u_{1002} \\ u_{1005} \end{matrix}$$

- Solve $\{\mathbf{K}_{FF} - \omega^2 \mathbf{M}_{ff}\} \{\phi_f\}$ which gives $\omega^2 = 0.1206, 1.0000, 2.3473, 3.5321$.

$$\phi_f = \left\{ \begin{matrix} 0.42850 & -0.5773 & -0.2280 & 0.6565 \\ 0.23150 & 1.0937 & 0.3188 & -0.8619 \\ -0.00572 & 0.0986 & 0.5464 & 0.7012 \\ 0.01370 & -0.2887 & 0.7705 & -0.7568 \end{matrix} \right\}$$

- Data recovery (grid point displacement for mode 1)

$$\text{Residual Structure } \phi = \begin{Bmatrix} 0 \\ 0.4285 \end{Bmatrix} \begin{matrix} u_1 \\ u_3 \end{matrix}$$

Solution by Hand (cont.)

- Superelement 2**

$$\text{for exterior points } \phi_{2G} = \begin{Bmatrix} 0 \\ 0.4285 \\ 0.0137 \end{Bmatrix} \begin{Bmatrix} u_1 \\ u_3 \\ u_{1005} \end{Bmatrix}$$

$$\phi_{21} = \{\phi_{G2}\} \{\phi_{2G}\} = \begin{bmatrix} 1.0 & 0 & 0 \\ 0.5 & 0.5 & 1.0 \\ 0 & 1.0 & 0 \end{bmatrix} \begin{Bmatrix} 0 \\ 0.4285 \\ 0.0137 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0.2280 \\ 0.4285 \end{Bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix}$$

- Superelement 1**

$$\text{for exterior points } \phi_{1G} = \begin{Bmatrix} 0.4285 \\ 0.2315 \\ -0.00572 \end{Bmatrix} \begin{Bmatrix} u_3 \\ u_{1001} \\ u_{1002} \end{Bmatrix}$$

$$\phi_{11} = \{\phi_{G1}\} \{\phi_{1G}\} = \begin{bmatrix} 1.0 & 0 & 0 \\ 1.0 & 0.6180 & -1.0 \\ 1.0 & 1.0 & 0.6180 \end{bmatrix} \begin{Bmatrix} 0.4285 \\ 0.2315 \\ -0.00572 \end{Bmatrix} = \begin{Bmatrix} 0.4285 \\ 0.5773 \\ 0.6565 \end{Bmatrix} \begin{Bmatrix} u_3 \\ u_4 \\ u_5 \end{Bmatrix}$$

Solution Using MSC Nastran SOL103

- The following input data was created and run in MSC Nastran:

```
$
$   sample_part_se.dat
$
SOL 103
CEND
TITLE = SAMPLE PROBLEM FOR CMS USING
PARTS
SPC = 1
DISP = ALL
PARAM,GRDPNT,0
PARAM,USETPRT,0
$
SUBCASE 1
  LABEL = CMS FOR PART 1
  SUPER = 1
  METHOD=1
$
SUBCASE 2
  LABEL = CMS FOR PART 2
  SUPER = 2
  METHOD = 2
$
SUBCASE 100
  LABEL = Residual MODES
  SUPER = 0
  METHOD = 100
$
BEGIN BULK
$
grid,3,,20.
conm2,13,3,,1.0
$
EIGRL,100,,4
```

```
BEGIN SUPER = 1
$
EIGRL,1,,,2
grid,3,,20.
grid,4,,30.
grid,5,,40.
spoint,101,102
qset1,0,101,102
$
CELAS2,3,1.,3,1,4,1
CELAS2,4,1.,4,1,5,1
CONM2,14,4,,1.
CONM2,15,5,,1.
$
BEGIN SUPER = 2
$
EIGRL,2,,,1
grid,1,,0.
grid,2,,10.
grid,3,,20.
spoint,201,202
qset1,0,201,202
$
CELAS2,1,1.,1,1,2,1
CELAS2,2,1.,2,1,3,1
CONM2,11,1,,1.
CONM2,12,2,,1.
SPC1,1,123456,1
$
ENDDATA
```

Output From Spring Model CMS Run

```
0          CMS FOR PART 1                                     SUPERELEMENT 1
                                                                SUBCASE 1

                                R E A L   E I G E N V A L U E S
                                (BEFORE AUGMENTATION OF RESIDUAL VECTORS)

MODE      EXTRACTION      EIGENVALUE      RADIANS      CYCLES      GENERALIZED      GENERALIZED
NO.        ORDER                                     MASS          STIFFNESS

    1         1      3.819660E-01      6.180340E-01      9.836316E-02      1.000000E+00      3.819660E-01
    2         2      2.618034E+00      1.618034E+00      2.575181E-01      1.000000E+00      2.618034E+00
```

Output From Spring Model CMS Run (cont.)

					SUPERELEMENT 2	
0	CMS FOR PART 2				SUBCASE 2	
REAL EIGENVALUES						
(BEFORE AUGMENTATION OF RESIDUAL VECTORS)						
MODE	EXTRACTION	EIGENVALUE	RADIANS	CYCLES	GENERALIZED	GENERALIZED
NO.	ORDER				MASS	STIFFNESS
1	1	2.000000E+00	1.414214E+00	2.250791E-01	1.000000E+00	2.000000E+00

Output From Spring Model CMS Run (cont.)

SUPERELEMENT 0						
0		RESIDUAL MODES			SUBCASE 100	
R E A L E I G E N V A L U E S						
MODE	EXTRACTION	EIGENVALUE	RADIANS	CYCLES	GENERALIZED	GENERALIZED
NO.	ORDER				MASS	STIFFNESS
1	1	1.206148E-01	3.472964E-01	5.527393E-02	1.000000E+00	1.206148E-01
2	2	1.000000E+00	1.000000E+00	1.591549E-01	1.000000E+00	1.000000E+00
3	3	2.347296E+00	1.532089E+00	2.438395E-01	1.000000E+00	2.347296E+00
4	4	3.532089E+00	1.879385E+00	2.991135E-01	1.000000E+00	3.532089E+00

Output From Spring Model CMS Run (cont.)

```

DATA RECOVERY FOR SUPERELEMENT      0 IS NOW INITIATED.
1  SAMPLE PROBLEM FOR CMS USING PARTS                                DECEMBER 21, 2011  MSC.NASTRAN 11/25/11  PAGE    58
                                                                    SUPERELEMENT 0
0  RESIDUAL MODES
  EIGENVALUE = 1.206148E-01
  CYCLES = 5.527393E-02      R E A L  E I G E N V E C T O R  N O .      1

  POINT ID.  TYPE      T1      T2      T3      R1      R2      R3
           3    G      4.285251E-01  0.0      0.0      0.0      0.0      0.0
1  SAMPLE PROBLEM FOR CMS USING PARTS                                DECEMBER 21, 2011  MSC.NASTRAN 11/25/11  PAGE    59
                                                                    SUPERELEMENT 0
0  RESIDUAL MODES
  EIGENVALUE = 1.000000E+00
  CYCLES = 1.591549E-01      R E A L  E I G E N V E C T O R  N O .      2

  POINT ID.  TYPE      T1      T2      T3      R1      R2      R3
           3    G      5.773503E-01  0.0      0.0      0.0      0.0      0.0
1  SAMPLE PROBLEM FOR CMS USING PARTS                                DECEMBER 21, 2011  MSC.NASTRAN 11/25/11  PAGE    60
                                                                    SUPERELEMENT 0
0  RESIDUAL MODES
  EIGENVALUE = 2.347296E+00
  CYCLES = 2.438395E-01      R E A L  E I G E N V E C T O R  N O .      3

  POINT ID.  TYPE      T1      T2      T3      R1      R2      R3
           3    G     -2.280134E-01  0.0      0.0      0.0      0.0      0.0
1  SAMPLE PROBLEM FOR CMS USING PARTS                                DECEMBER 21, 2011  MSC.NASTRAN 11/25/11  PAGE    61
                                                                    SUPERELEMENT 0
0  RESIDUAL MODES
  EIGENVALUE = 3.532089E+00
  CYCLES = 2.991135E-01      R E A L  E I G E N V E C T O R  N O .      4

  POINT ID.  TYPE      T1      T2      T3      R1      R2      R3
           3    G      6.565385E-01  0.0      0.0      0.0      0.0      0.0

```

Output From Spring Model CMS Run (cont.)

```

DATA RECOVERY FOR SUPERELEMENT 1 IS NOW INITIATED.
1  SAMPLE PROBLEM FOR CMS USING PARTS                                DECEMBER 21, 2011  MSC.NASTRAN 11/25/11  PAGE 63
                                                                    SUPERELEMENT 1
0  CMS FOR PART 1                                                    SUBCASE 1
  EIGENVALUE = 1.206148E-01
  CYCLES = 5.527393E-02      R E A L  E I G E N V E C T O R  N O .      1

  POINT ID.  TYPE      T1      T2      T3      R1      R2      R3
    3        G      4.285251E-01  0.0      0.0      0.0      0.0      0.0
    4        G      5.773503E-01  0.0      0.0      0.0      0.0      0.0
    5        G      6.565385E-01  0.0      0.0      0.0      0.0      0.0
   101        S      2.722018E-01 -6.724520E-03
                                                                    SUPERELEMENT 1
0  CMS FOR PART 1                                                    SUBCASE 1
  EIGENVALUE = 1.000000E+00
  CYCLES = 1.591549E-01      R E A L  E I G E N V E C T O R  N O .      2

  POINT ID.  TYPE      T1      T2      T3      R1      R2      R3
    3        G      5.773503E-01  0.0      0.0      0.0      0.0      0.0
    4        G     -4.440892E-16  0.0      0.0      0.0      0.0      0.0
    5        G     -5.773503E-01  0.0      0.0      0.0      0.0      0.0
   101        S     -1.285778E+00 -1.159385E-01
                                                                    SUPERELEMENT 1
0  CMS FOR PART 1                                                    SUBCASE 1
  EIGENVALUE = 2.347296E+00
  CYCLES = 2.438395E-01      R E A L  E I G E N V E C T O R  N O .      3

  POINT ID.  TYPE      T1      T2      T3      R1      R2      R3
    3        G     -2.280134E-01  0.0      0.0      0.0      0.0      0.0
    4        G     -5.773503E-01  0.0      0.0      0.0      0.0      0.0
    5        G      4.285251E-01  0.0      0.0      0.0      0.0      0.0
   101        S      3.748278E-01  6.423264E-01
                                                                    SUPERELEMENT 1
0  CMS FOR PART 1                                                    SUBCASE 1
  EIGENVALUE = 3.532089E+00
  CYCLES = 2.991135E-01      R E A L  E I G E N V E C T O R  N O .      4

  POINT ID.  TYPE      T1      T2      T3      R1      R2      R3
    3        G      6.565385E-01  0.0      0.0      0.0      0.0      0.0
    4        G     -5.773503E-01  0.0      0.0      0.0      0.0      0.0
    5        G      2.280134E-01  0.0      0.0      0.0      0.0      0.0
   101        S     -1.013219E+00  8.243195E-01
  
```

Output From Spring Model CMS Run (cont.)

```

DATA RECOVERY FOR SUPERELEMENT      2 IS NOW INITIATED.
1  SAMPLE PROBLEM FOR CMS USING PARTS                                DECEMBER 21, 2011  MSC.NASTRAN 11/25/11  PAGE    68
                                                                    SUPERELEMENT 2
                                                                    SUBCASE 2
0  CMS FOR PART 2
  EIGENVALUE = 1.206148E-01
  CYCLES = 5.527393E-02      R E A L  E I G E N V E C T O R  N O .      1

  POINT ID.  TYPE      T1      T2      T3      R1      R2      R3
    1        G        0.0      0.0      0.0      0.0      0.0      0.0
    2        G      2.280134E-01  0.0      0.0      0.0      0.0      0.0
    3        G      4.285251E-01  0.0      0.0      0.0      0.0      0.0
   201        S      1.375089E-02  0.0
                                                                    SUPERELEMENT 2
                                                                    SUBCASE 2
0  CMS FOR PART 2
  EIGENVALUE = 1.000000E+00
  CYCLES = 1.591549E-01      R E A L  E I G E N V E C T O R  N O .      2

  POINT ID.  TYPE      T1      T2      T3      R1      R2      R3
    1        G        0.0      0.0      0.0      0.0      0.0      0.0
    2        G      5.773503E-01  0.0      0.0      0.0      0.0      0.0
    3        G      5.773503E-01  0.0      0.0      0.0      0.0      0.0
   201        S      2.886751E-01  0.0
                                                                    SUPERELEMENT 2
                                                                    SUBCASE 2
0  CMS FOR PART 2
  EIGENVALUE = 2.347296E+00
  CYCLES = 2.438395E-01      R E A L  E I G E N V E C T O R  N O .      3

  POINT ID.  TYPE      T1      T2      T3      R1      R2      R3
    1        G        0.0      0.0      0.0      0.0      0.0      0.0
    2        G      6.565385E-01  0.0      0.0      0.0      0.0      0.0
    3        G     -2.280134E-01  0.0      0.0      0.0      0.0      0.0
   201        S      7.705452E-01  0.0
                                                                    SUPERELEMENT 2
                                                                    SUBCASE 2
0  CMS FOR PART 2
  EIGENVALUE = 3.532089E+00
  CYCLES = 2.991135E-01      R E A L  E I G E N V E C T O R  N O .      4

  POINT ID.  TYPE      T1      T2      T3      R1      R2      R3
    1        G        0.0      0.0      0.0      0.0      0.0      0.0
    2        G     -4.285251E-01  0.0      0.0      0.0      0.0      0.0
    3        G      6.565385E-01  0.0      0.0      0.0      0.0      0.0
   201        S     -7.567943E-01  0.0

```

WORKSHOP 7

Normal Modes Analysis using Parts Superelement

SECTION 13

Rotordynamics

Introduction

- **Examples of Rotating Structures**
 - Jet Engines
 - Turbines
 - Centrifuge
 - Compressors
 - Energy Storage Devices
 - Motors
 - ...

Introduction (cont.)

- **Why Be Concerned about Rotordynamics?**
 - Apply predictive methods for dynamic response characteristics to improve operation of structures
 - Rotating structures contain large amounts of kinetic energy, failures cause extensive structural damage and can be in some cases catastrophic
 - Behavior is not intuitive, e.g., gyroscopic effects, forces acting in one direction cause reactions in other directions

Overview of Rotordynamics

- **Types of analyses**
 - Static analysis
 - SOL 101
 - Whirl modes, Campbell diagrams
 - SOL 107 (direct) or SOL 110 (modal)
 - Critical speed prediction
 - SOL 107 (direct) or SOL 110 (modal)
 - Frequency response
 - SOL 108 (direct) or SOL 111 (modal)
 - Direct Transient response
 - SOL 109 (linear) or SOL 129, 400 (nonlinear)
- **A dynamic solution is needed for most rotordynamic analyses, e.g., aircraft maneuver condition like a abort pull-up**
- **Special cases may be solved with static analysis, e.g., aircraft in a steady turn**

Overview of Rotordynamics (cont.)

- **Assumptions and Limitations**

- Analysis performed in a stationary (inertial) coordinate system, i.e., non-rotating
- Models must be axisymmetric, e.g, cyclic symmetric with 3 or more segments
- Center-line model, boundary grids must be on the center-line
 - Use SE Guyan reduction for 3D models
- Connect rotor models to support structure by rigid elements only, elastic coupling at the g-set is not allowed unless ROTORSE option is used for 3D model reduction.
- Rotor axis is flexible, disks are rigid
- Results correct for only the axis of rotation

- **Critical speeds and modes only available for the reference rotor**

- **Modes valid between SPDLOW and SPDHIGH specified on RGYRO entry**

Damping Options

- **Damping appears in the equations of motion as either viscous or structural damping**
 - Viscous damping are real matrix terms applied to the damping matrix multiplying the velocity vector
 - Structural damping are imaginary terms applied to the stiffness matrix multiplying the displacement vector

Damping Options (cont.)

- **MSC Nastran offers multiple methods of inputting the two forms of damping**
 - Structural damping derives from using parameter G, the GE field on the MAT1 and some property entries, or direct matrix input of K2GG and K2PP matrices
 - Viscous damping derives from entries such as CVISC, CDAMP, CBUSH, or from proportional (sometimes referred to as Rayleigh) damping using parameters ALPHA1 and ALPHA2, or from direct matrix input of B2GG and B2PP matrices
 - Transient solutions require real matrix terms so structural damping is converted to equivalent viscous damping requiring parameters W3 and W4.
- **Modal damping derives from TABDMP1 entries requested by SDAMPING case control requests.**

Damping Options (cont.)

- The comments of the preceding slide apply primarily to more commonly used input for the stationary structural components.
- An alternate form of input for the stationary structure (this form does NOT apply to the rotating structural components) will be discussed in more detail in the input description following this introduction to damping.
- The alternate form of damping input overrides the parameter specified on parameter G, W3, W4 and WH. The alternate form of input offers a hybrid (modal) damping input for stationary structural components in the direct transient solutions.
- Damping applied to the rotating structural components derives from input specified on RSPINR and RSPINT bulk data entries with the structural damping coefficient input on GE field of the MATi entry.
- Damping applied to the rotating structural components uses similar parameter input and contains a hybrid damping for direct solutions using modal damping input.

New Forms of DAMPING Input

- ***Nearly all*** damping input specified by the user is consolidated into one input format.
- Different forms of damping are now case control command/bulk data entry selectable consolidating the use of parameters, G, ALPHA1, ALPHA2, W3, W4, WH and GE material scaling
- **Case Control**
 - SEDAMP
 - RSDAMP
- **Bulk Data**
 - DAMPING
 - HYBDAMP

New Forms of DAMPING Input (cont.)

- **SEDAMP and RSDAMP Case Control Commands**
 - SEDAMP
 - Requests parameter and hybrid damping for superelements
 - SEDMAP = n
 - Where n is identification number of a DAMPING bulk data entry
 - RSDAMP
 - Requests parameter and hybrid damping for the residual structure
 - RSDAMP = n
 - Where n is the identification number of a DAMPING bulk data entry

Additional Damping Options – HYBDAMP

- Hybrid modal damping for direct dynamic solutions
- Specifies the eigenvalue extraction method and damping for hybrid damping calculations.
- Format

1	2	3	4	5	6	7	8	9	10
HYBDAMP	ID	METHOD	SDAMP	KDAMP					

- Example

1	2	3	4	5	6	7	8	9	10
HYBDAMP	1	100	200						

Additional Damping Options – HYBDAMP

<u>Field</u>	<u>Contents</u>
ID	Identification number of HYBDAMP entry (Integer > 0; Required)
METHOD	Identification number of METHOD entry for modes calculation (Integer > 0; Required)
SDAMP	Identification number of SDAMP entry for modes calculation (Integer > 0; Required)
KDAMP	Selects modal “structural” damping (Character: “YES or “NO”, see Remark 1; Default = “NO”)

Rotordynamics Bulk Data Entries

Table of Rotordynamic Entries versus Analysis Discipline

Entry	Static	Complex Eigenvalue	Frequency Response	Transient Response
ROTORG	*	*	*	*
RGYRO	*	*	*	
RSPINR	*	*	*	
RSPINT				*
UNBALNC			* (optional)	* (optional)
ROTORSE	*	*	*	*

Rotordynamics Bulk Data Entries (cont.)

- **RGYRO**—specifies synchronous or asynchronous analysis, and rotation speed of the reference rotor and reference rotor ID

Format:

RGYRO	RID	SYNCFLG	REFROTR	SPDUNIT	SPDLOW	SPDHIGH	SPEED		

Example:

RGYRO	100	ASync	1	RPM			2000.		

Rotordynamics Bulk Data Entries(cont.)

- **ROTORG**—specifies grid points that compose the rotor line model

Format:

ROTORG	ROTORID	GRID1	GRID2	GRID3	...	GRIDn			

or

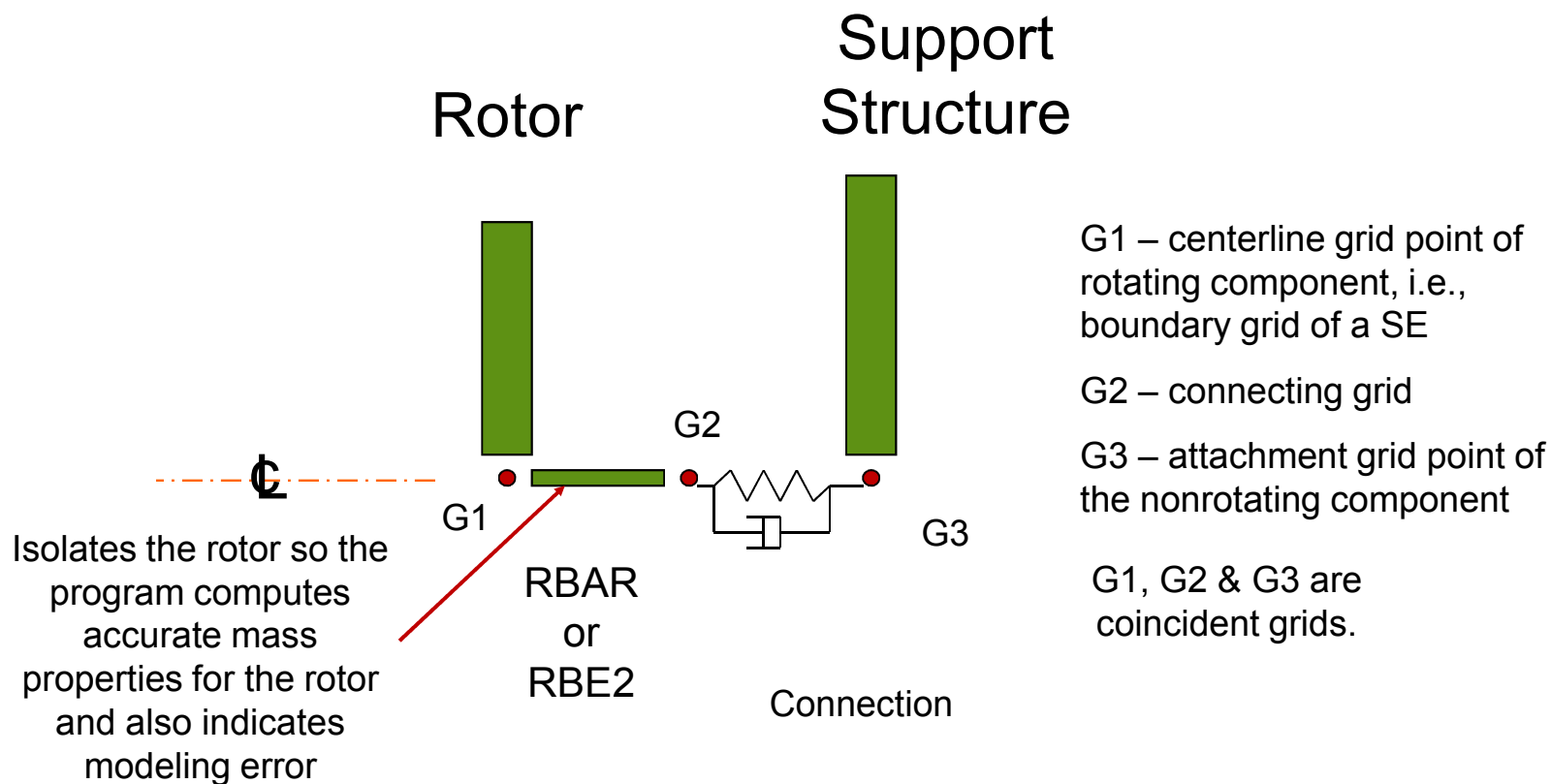
ROTORG	ROTORID	GRID1	THRU	GRID2	BY	Inc			

Example:

ROTORG	1	1	THRU	101	BY	10			

Connection for Rotor and Support Structure

- Schematic Example of Connection when using ROTORG



- G2 & RBAR/RBE2 not needed with ROTORSE

Comments

- **Proper Rotor/Structure Connection avoids adding miscellaneous mass to the rotor and circulation damping terms caused by support structure stiffness.**
- **Note that the dependent/independent dofs of the RBAR or RBE2 does not matter since the rotor mass and circulation damping are based on the g-set dofs.**
- **ROTORSE changes the above rules**

Rotordynamics Bulk Data Entries(cont.)

- **ROTORSE Specifies grids that compose the rotor line model**
 - The boundary grids for a rotor specified with the ROTORSE in place of the ROTORG must still follow the same rules as the ROTORG input.

Format:

ROTORSE	ROTORID	SEID	SEOPT						

Example:

ROTORSE	10	1							

Rotordynamics Bulk Data Entries (cont.)

- **RSPINR**—specifies the relative spin rates between rotors for complex eigenvalue, frequency response, and static analysis
 - Also defines positive rotor spin direction (GA to GB)

Format:

RSPINR	ROTORID	GRIDA	GRIDB	SPDUNIT	SPTID				
	GR	ALPHAR1	ALPHAR2	HYBRID					

Example:

RSPINR	100	1001	1002	RPM	100				
	0.02			1001					

Rotordynamics Bulk Data Entries (cont.)

- **RSPINT**—specifies rotor spin rates for transient analysis
 - Also defines positive rotor spin direction (right hand rule going from GA to GB)

Format:

RSPINT	ROTORID	GRIDA	GRIDB	SPDUNIT	SPTID	SPDOUT			
	GR	ALPHAR1	ALPHAR2	HYBRID					

Example:

RSPINT	100	1001	1002	RPM	1001				
	0.02	0.01	0.002						

Rotordynamics Bulk Data Entries (cont.)

- UNBALNC**—specifies unbalance load for transient or frequency response analysis defined in a cylindrical coordinate system with the rotor rotational axis as the z-axis

Format:

UNBALNC	RID	MASS	GRID	X1	X2	X3			
	ROFFSET	THETA	ZOFFSET	T/Fon	T/Foff				
	UFT1	UFT2	UFT3	UFR1	UFR2	UFR3			
	MCT1	MCT2	MCT3	MCR1	MCR2	MCR3			
	SCR1	SCR2	SCR2						

Example:

UNBALNC	100	.1	1001	0.0	1.0	0.0			
	0.02	30.0	0.5			-1			

Parameters

Four parameters added for the rotor dynamics capability

- PARAM,GYROAVG,x (default=0)
 - If x=-1, the gyroscopic terms are generated using a least square fit of terms within the analysis range
- PARAM,WR3,x; PARAM,WR4,z, and PARAM,WRH,z
 - Specifies “average” excitation for calculation of rotor damping and circulation terms
 - This is similar to param,w3,y and param,w4,z in transient analysis

Examples Cases

- **Dimentberg Rotor**
 - Whirl frequencies
 - Critical speeds
 - Frequency response analysis
 - Linear transient response analysis
 - Static analysis

Dimentberg Example Shaft and Rigid Disk*

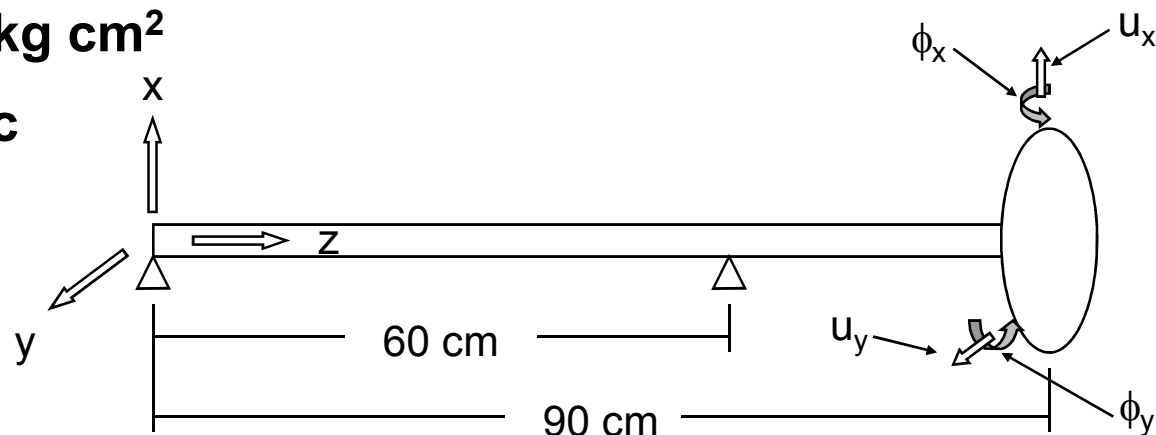
$$MSC = 0.0157 \text{ kg sec}^2/\text{cm}$$

$$I_d = 2.45 \text{ kg/sec}^2 \text{ cm}$$

$$I_p = 2 I_d$$

$$EI = 1,647,700 \text{ kg cm}^2$$

$$\Omega = 100 \text{ rad/sec}$$



*References: Bedrossian, H., and Viekos, N., Rotor-Disk System Gyroscopic Effects in MSC/NASTRAN Dynamics Solutions, MD.NASTRAN User's Conf. Proc., Paper No. 12, 1982.

Dimentberg, F. M., *Flexural Vibrations of Rotating Shafts*, Butterworths, London, 1964

Rotordynamic Matrix Terms at One Point

- Matrix Terms for at One Point with a Constant Spin Speed, Ω , ASYNC on RSPINR

6x6 Damping Matrix

$$M_{u_x u_x} \ddot{u}_x + K_{u_x u_x} u_x - K_{u_x \phi_y} \phi_y = 0$$

$$M_{u_y u_y} \ddot{u}_y + K_{u_y u_y} u_y + K_{u_y \phi_x} \phi_x = 0$$

$$I_d \ddot{\phi}_x + K_{\phi_x u_y} u_y + K_{\phi_x \phi_x} \phi_x + I_p \Omega \dot{\phi}_y = 0$$

$$I_d \ddot{\phi}_y - K_{\phi_y u_x} u_x + K_{\phi_y \phi_y} \phi_y - I_p \Omega \dot{\phi}_x = 0$$

$$\begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & I_p \Omega & 0 \\ 0 & 0 & 0 & -I_p \Omega & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}$$

---- eliminate 3rd and 6th row and column to form 4x4 matrix

Rotordynamic Matrix Terms at One Point

- Matrix Terms for at One Centerline Point with Rotor Spin Speed equal to the Excite or Eigenvalue Frequency, ω , **SYNC**

6x6 Mass Matrix

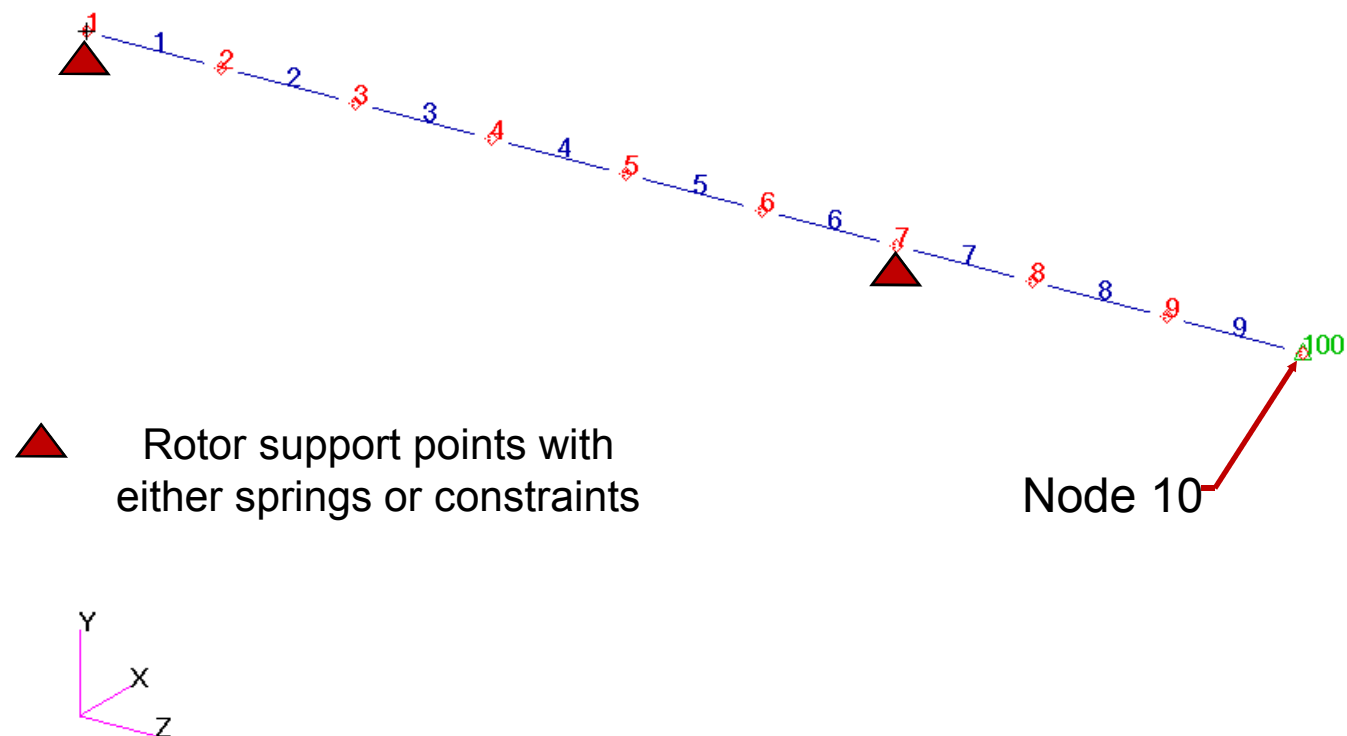
$$\begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & i\Omega & 0 \\ 0 & 0 & 0 & -i\Omega & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}$$

Complex Eigenvalue Analysis

- **Whirl Frequencies**
 - Beam model setup with DMIG gyroscopic coupling
 - Beam model RGYRO setup without superelements
 - 3D model with a superelement
- **Critical Speeds**

Line Model w/o Superelements

- **CBAR Elements with CONM2 100 at Node 10**



Line Model (cont.)

- **Is it possible to include rotordynamics effects without the use of capability and no DMAP alters?**
- **The answer is YES!**
- **But there is a price**
- **The next slide illustrates what is needed**

Example Shaft and Disk, DMIG Setup

ID ROTATING DISK

SOL 107

CEND

TITLE = GYROSCOPIC INFLUENCE OF A RIGID DISK ROTATING ON A SHAFT

SUBTI = NEARLY MASSLESS SHAFT, SPIN RATE OF 100.0 RAD/SEC

B2PP = GYROD

SPC = 1

CMETHOD = 1

DISP(PHASE) = ALL

BEGIN BULK

.

\$ DISK MASS AND GYRO SPECIFICATIONS

CONM2 100 10 157.0-4

2.45 2.45

\$dmig name "0" ifo tin tout polar ncol

DMIG GYROD 0 1 1 0

DMIG GYROD 10 4 10 5

DMIG GYROD 10 5 10 4

\$ COMPLEX EIGENVALUE EXTRACTION

EIGC 1 HESS MAX

ENDDATA

Note: I_p is not needed on CONM2 unless torsion modes are to be calculated

Value is ΩI_p

-490.0
490.0

Example Shaft and Disk, RGYRO Setup

ID ROTATING DISK

SOL 107

CEND

TITLE = GYROSCOPIC INFLUENCE OF A RIGID DISK ROTATING ON A SHAFT

SUBTI = NEARLY MASSLESS SHAFT, SPIN RATE OF 100.0 RAD/SEC

SPC = 1

RGYRO = 1

CMETHOD = 1

DISP(PHASE) = ALL

BEGIN BULK

.

\$ DISK MASS AND GYRO SPECIFICATIONS

CONM2 100 10 157.0-4

2.45 2.45

\$ GYROSCOPIC COUPLING AND SPEED CONTROL

\$rotorg rotorid gid1 gid2 etc

ROTORG 1 1 thru 10 by 1

\$rgyro rid syncflg refrotr spdunit spdlow spdhigh speed

RGYRO 1 ASYNC 1 RPM 954.93

\$rspinr rotorid grida gridb spdunit sptid

RSPINR 1 9 10 RPM 954.93

\$ COMPLEX EIGENVALUE EXTRACTION

EIGC 1 HESS MAX 8

ENDDATA

Note: Multiple SUBCASEs are allowed to run different speeds on the selected RGYRO entry

Note: I_p is required on the CONM2

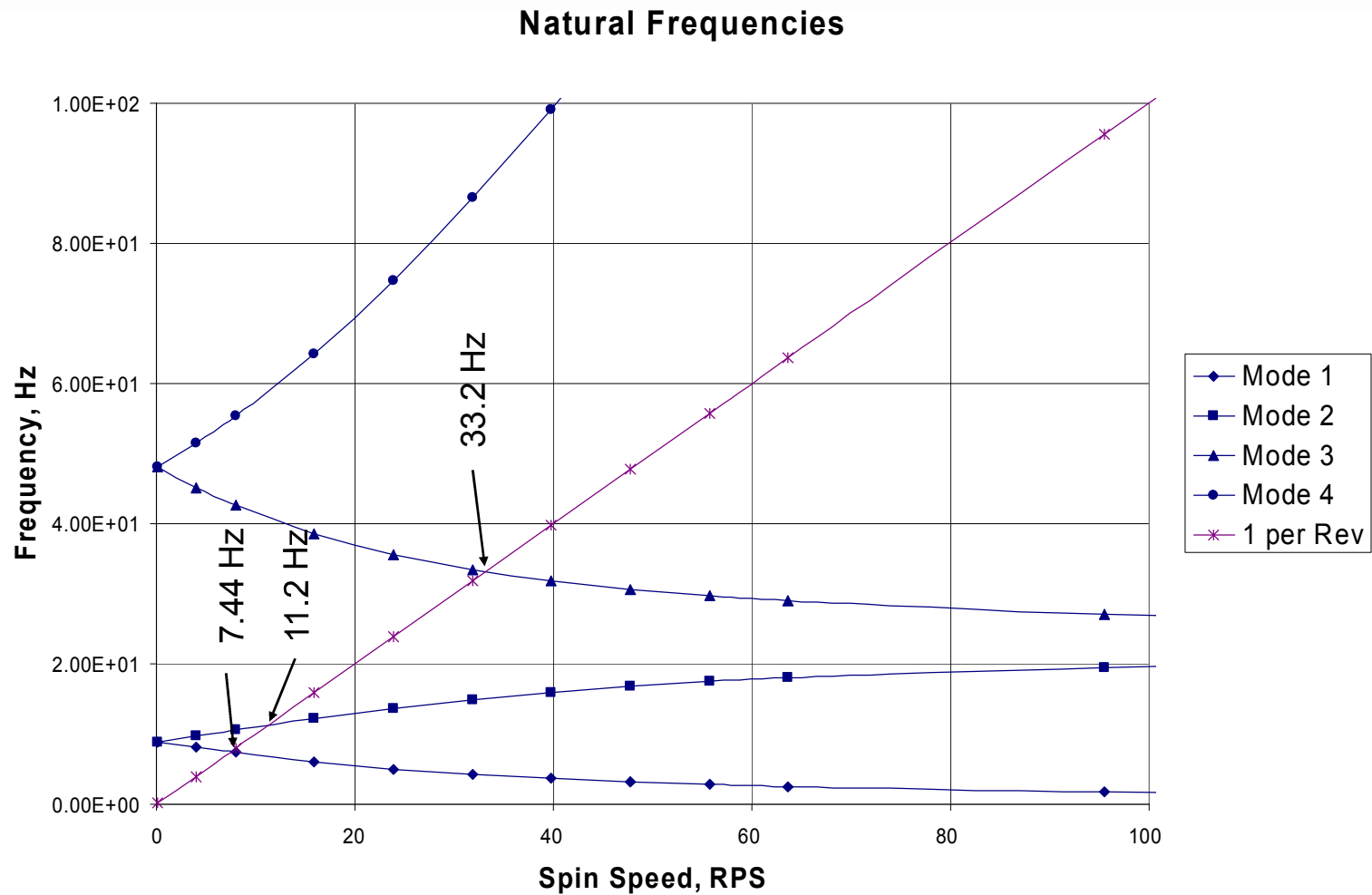
Combined to compute ΩI_p

Results of Example Shaft and Disk, RGYRO or DMIG Yield Same Eigenvalues

C O M P L E X E I G E N V A L U E S U M M A R Y					
ROOT	EXTRACTION	EIGENVALUE		FREQUENCY	DAMPING
NO.	ORDER	(REAL)	(IMAG)	(CYCLES)	COEFFICIENT
1	2	7.204462E-15	-3.805280E+01	6.056291E+00	-3.786561E-16
2	1	7.204462E-15	3.805280E+01	6.056291E+00	-3.786561E-16
3	4	-2.242220E-14	-7.656962E+01	1.218643E+01	5.856683E-16
4	3	-2.242220E-14	7.656962E+01	1.218643E+01	5.856683E-16
5	6	4.939756E-14	-2.423585E+02	3.857254E+01	-4.076405E-16
6	5	4.939756E-14	2.423585E+02	3.857254E+01	-4.076405E-16
7	8	2.961827E-14	-4.038409E+02	6.427328E+01	-1.466829E-16
8	7	2.961827E-14	4.038409E+02	6.427328E+01	-1.466829E-16

- Use Eigenvectors from the Eigenvalue Table with the Positive Imaginary Part

Campbell Diagram – Non-SE Model



Example Critical Speed Setup

ID ROTATING DISK

SOL 107

CEND

TITLE = GYROSCOPIC INFLUENCE OF A RIGID DISK ROTATING ON A SHAFT,

SUBTI = NEARLY MASSLESS SHAFT, CRITICAL SPEED ANALYSIS

SPC = 1

RGYRO = 1

CMETHOD = 1

DISP(PHASE) = ALL

BEGIN BULK

.

\$ DISK MASS AND GYRO SPECIFICATIONS

CONM2 100 10 157.0-4

2.45 2.45

4.9

Changed from ASYNC

\$ GYROSCOPIC COUPLING AND SPEED CONTROL

\$rotorg rotorid gid1 gid2 etc

ROTORG 1 1 thru 10 by 1

\$rgyro rid syncflg refrotr spdunit spdlow spdhigh speed

RGYRO 1 SYNC 1 RPM 954.93

\$rspinr rotorid grida gridb spdunit sptid

RSPINR 1 9 10 RPM 954.93

\$ COMPLEX EIGENVALUE EXTRACTION

EIGC 1 HESS MAX 8

ENDDATA

Note: I_p is required on the CONM2

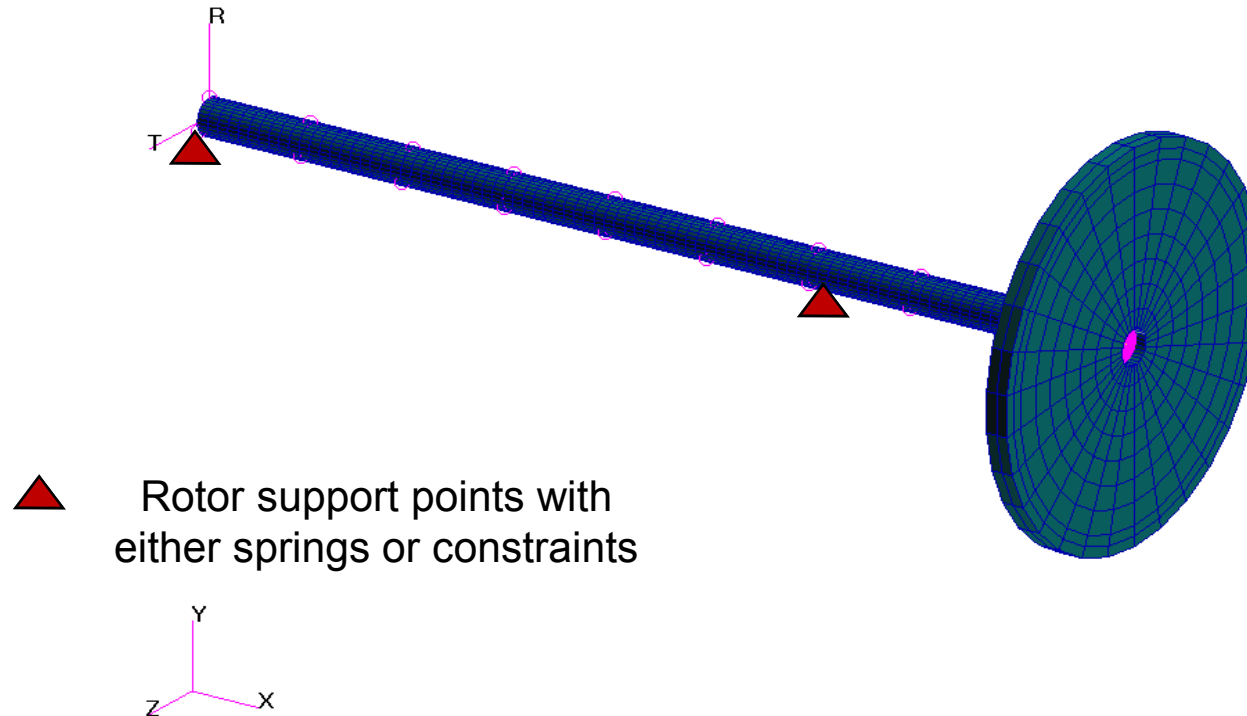
Results of Critical Speed Analysis

C O M P L E X E I G E N V A L U E S U M M A R Y					
ROOT	EXTRACTION	EIGENVALUE		FREQUENCY	DAMPING
NO.	ORDER	(REAL)	(IMAG)	(CYCLES)	COEFFICIENT
1	4	-5.323785E-14	4.676258E+01	7.442496E+00	2.276942E-15
2	3	4.162563E-16	7.063671E+01	1.124218E+01	-1.178583E-17
3	2	-1.070884E-15	2.084957E+02	3.318313E+01	1.027248E-17
4	1	2.390711E+02	1.472887E-15	0.0	0.0

3-D Example

3-D Shaft and Disk Model

- **QUAD4 Shaft, HEXA Disk, RBE3 to centerline**

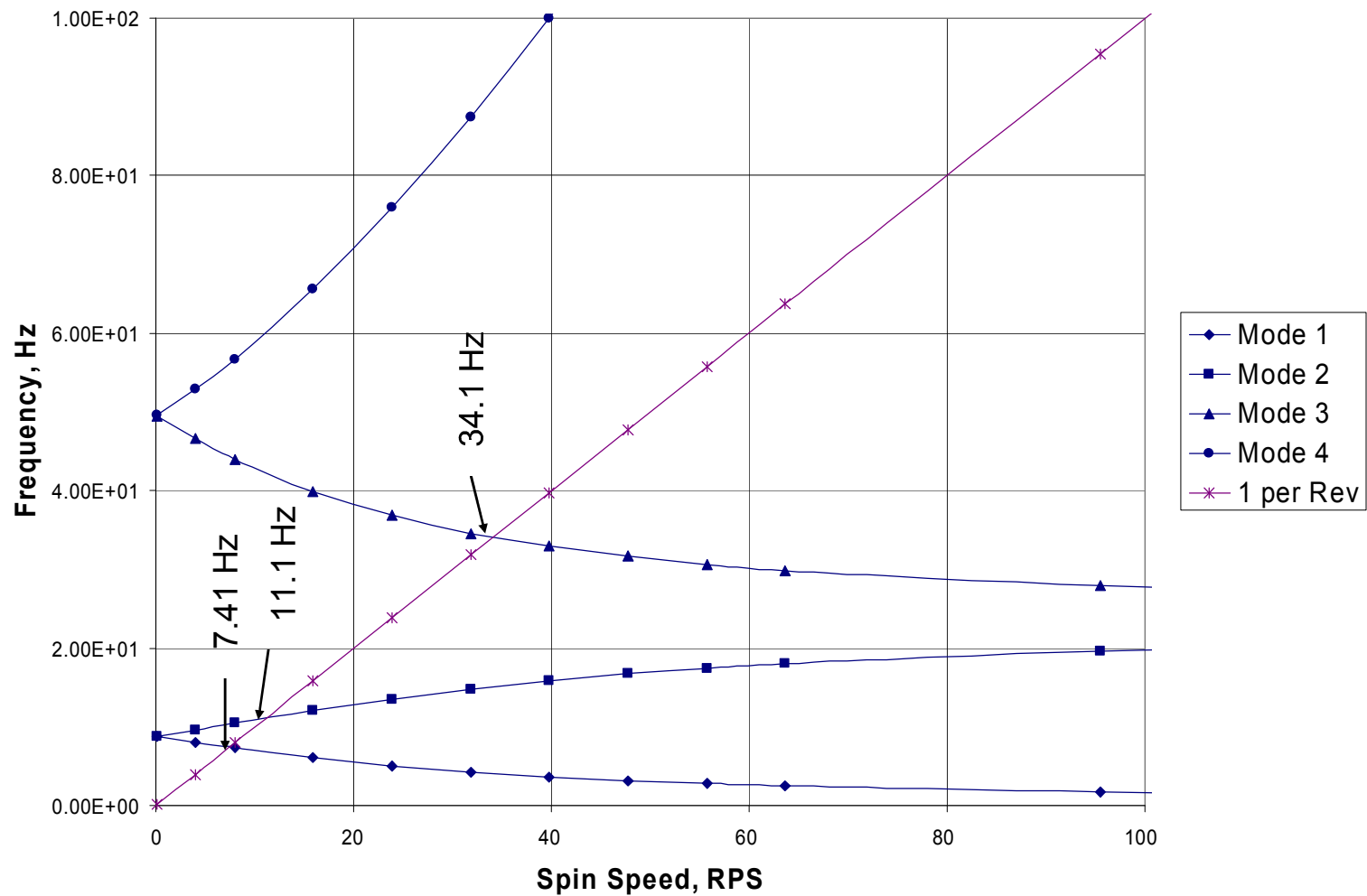


Example 3-D Shaft and Disk RGYRO Model

```
ID THREE-D MODEL
SOL 107
CEND
TITLE = QUAD4/HEXA MODEL OF A 3.6 CM DIA, 0.0976 CM THICK. SHAFT
SUBTI = 17.666 CM AVG Radius of Overhung Disk at free end (sta 90) of shaft
LABEL = Two support points at sta 0 and sta 60,
  CMETHOD = 1
  DISP(PLOT)      = ALL
$
SUBCASE 1
  LABEL = 15.915 RPS
  RGYRO = 1
.
BEGIN BULK
$rgyro  rid      syncflg  refrotr  spdunit  spdlow  spdhigh  speed
RGYRO   1        ASYNC    1         FREQ                    15.9155
$rotorg  rotorid  gid1     gid2     etc
ROTORG  1        1000     THRU     10000   by      1000
$rspinr  rotorid  grida    gridb    spdunit  sptid
RSPINR  1        9000     10000   FREQ     15.9155
$ Structural Model
```

Campbell Diagram – 3D Model w/ SE

Natural Frequencies



Nonlinear Frequency Response Analysis

- **SENLHARM is a nonlinear frequency response analysis allowing NOLINI and NLRGAP bulk data entry input**
 - SOL SENLHARM
- **New Case Control Command**
 - NLHARM
- **New Bulk Data Entries**
 - NLHARM
 - NLFREQ
 - NLFREQ1

SOL SENLHARM Case Control

- **NLHARM Case Control Command**
 - NLHARM = n
 - Where n is the id of a NLHARM bulk data entry

SOL SENLHARM Bulk Data Entries

- **NLHARM bulk data entry defines parameters nonlinear harmonic analysis with SOL 128**

- **Format:**

NLHARM	ID	SUBFAC	NHARM	NLFREQ					

- **Example:**

NLHARM	1000	1	2	100					

SOL SENLHARM Bulk Data Entries

- **NLFREQ bulk data entry defines a list of user supplied excitation frequencies for SOL 128**
- **Format:**

NLFREQ	ID	F1	F2	...	FN				

- **Example:**

NLFREQ	100	10.0	20.0	30.0	40.0	50.0			

Field

Contents

- ID Identification number
- Fi Frequency values in cycles per unit time

SOL SENLHARM Bulk Data Entries

- **NLFREQ1 bulk data entry defines a list of user supplied excitation frequencies for SOL 128**

- **Format:**

NLFREQ1	ID	F1	DF	NDF					

- **Example:**

NLFREQ1	100	0.5	0.5	59					

Nonlinear Frequency Response Analysis

- **SENLHARM, SOL 128, is a nonlinear frequency response analysis allowing NOLINI and NLRGAP bulk data entry input**
 - SOL 128
- **New Case Control Command**
 - NLHARM
- **New Bulk Data Entries**
 - NLHARM
 - NLFREQ
 - NLFREQ1

SECTION 14

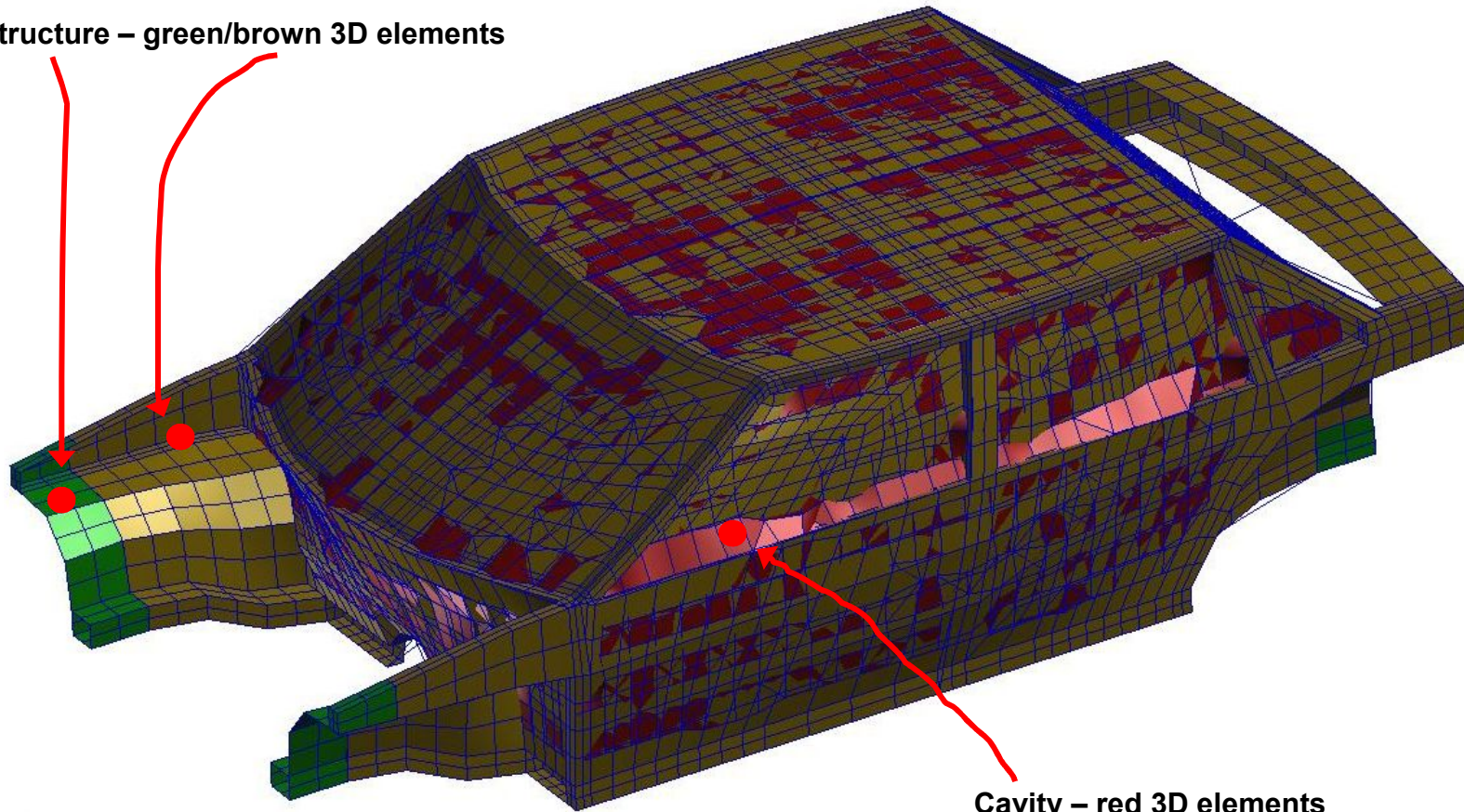
Acoustics

INTERIOR ACOUSTICS

Example Model

- *Structure and Cavity*

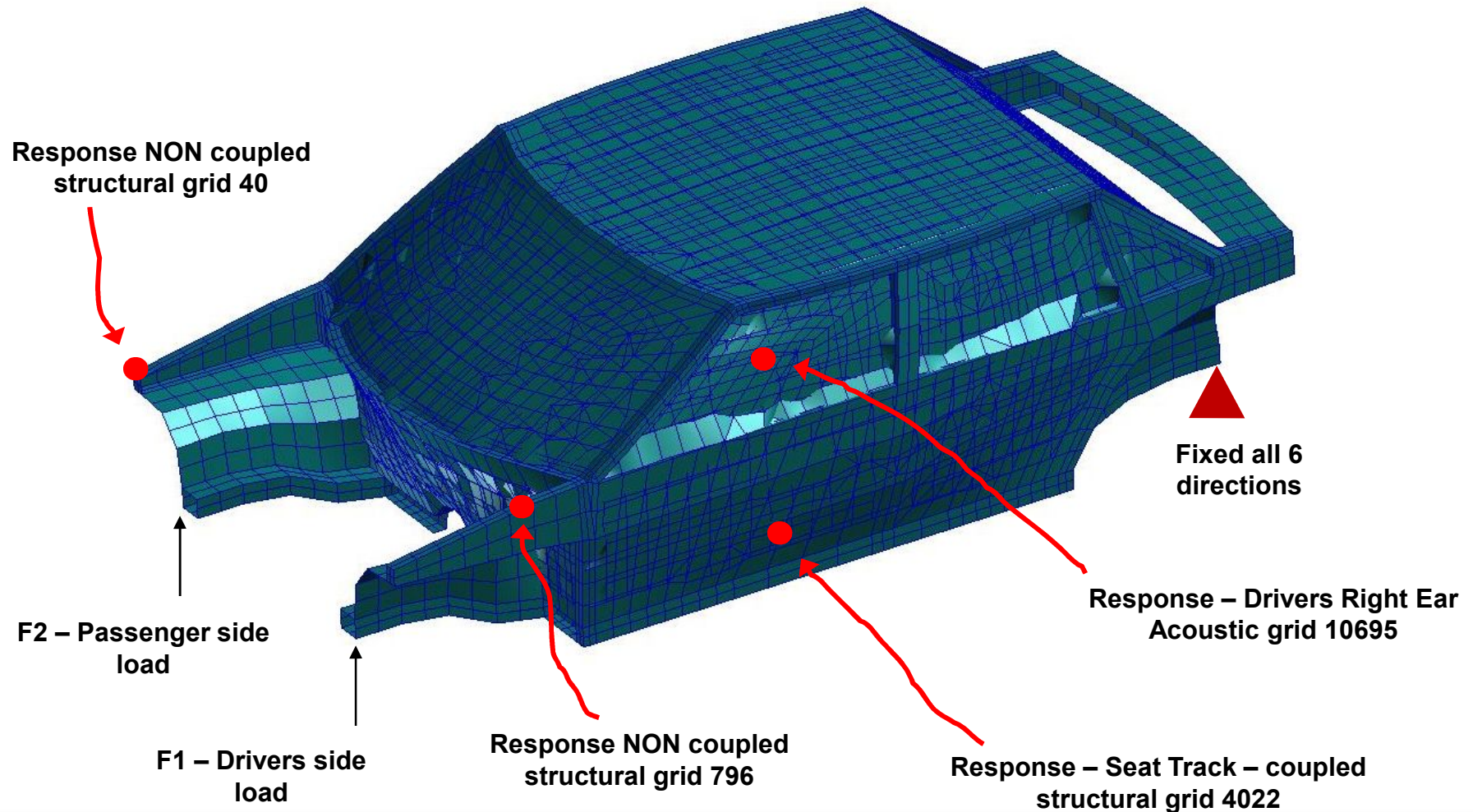
Structure – green/brown 3D elements



Cavity – red 3D elements

Example Model

- Loading, Constraints, Responses***



Introduction

- ***Fluid/Acoustic Options***
 - **3D Fluid/Acoustic Cavity**
 - **Most common used acoustic option**
 - **Example Applications**
 - **Car, Truck, or plane cabin**
 - **Finite element**
 - **Modeled with solid elements (HEXA/PENTA/TETRA)**
- **Fluid Structure Coupling**
 - **Driven by existing structure mesh**
 - **Fluid cavity mesh**
 - **Does not have to match grid point to grid point with structure**

3D Cavity Generation

- **Properties**
 - 3D solid elements
 - HEXA, PENTA, TETRA
 - PSOLID option FCTN set to PFLUID

PSOLID	PID	MID	CORDM	IN	STRESS	ISOP	FCTN		

- **MAT10**

MAT10	MID	BULK	RHO	C	GE				

$$\text{BULK} = C^2 * \text{RHO}$$

Coupling - Structure/Fluid Interaction

- **Three methods available**
 - BW (Default, recommended method)
 - CP
 - IDENT
- **Use the default ACMODL (BW) as a starting point**
 - No ACMODL needed
 - Additional options to improve coupling matrix
- **If manual intervention is needed**
 - Define ACMODL

Coupling - Structure/Fluid Interaction (cont.)

- Production models rarely have coincident fluid/structure grid points
- The DMAP module ACMG contains algorithms which will find the fluid/structure grid point pairs
 - Selection is done with the BULK DATA ACMODL entry

(METHOD="BW")

Default and preferred method

1	2	3	4	5	6	7	8	9	10
ACMODL	INTER	INFOR	FSET	SSET	NORMAL	METHOD	SKNEPS	DSKNEPS	
	INTOL	ALLSSET	SRCHUNIT						

(METHOD="CP")

Old method

ACMODL	INTER	INFOR	FSET	SSET	NORMAL	METHOD			

Coupling - Structure/Cavity Interaction

- ***ACMSCOL fluid structure grid point pair options***
 - User supplied as FSET/SSET defined by SET1
 - Tedious for production models
 - Automatic search methods
 - CP of Closed Pressure vessel
 - BW for Body in White
 - The selection of the search method also selects the calculation method of the coupling matrix
 - BW is the default and is the most robust search method for fluid structure grid point pairs
 - Each method implies calculation of the A matrix
 - BW A matrix $\neq$ CP A matrix

Coupling - Structure/Cavity Interaction

- **BW search method**
 - Introduced in V2004
 - Vast improvement in calculation time
 - CP method tends to ‘auger in’ with excessive CPU time
 - New searching methodology
 - param,skinout,punch produces .pch file that represent fluid “skin”
 - A utility program is provided for viewing and inspecting the completeness and accuracy of the “skin”

Coupling Matrix

- ***The A matrix***

- Represents the physical coupling of the cavity and structure
- Requires a list of surface fluid grids and closest structural grid
 - Production models rarely have coincident fluid/structure gridpoint locations
- Generated using the list of fluid/structure grid points
 - NASTRAN ACMG module
 - BW couples all 6 structural DOF

- ***Direct input of A matrix***

- This is controlled by the A2GG Case Control command
- The format of the A2GG input is similar to the other G-type DMIG input (e.g., K2GG)

Modal Space Solutions

- ***SOL Driven Mode Extraction***
 - Separate extraction of fluid and structure modes
 - Complex Modal space (SOL 110)
 - Modal frequency response (SOL 111) and modal transient response (SOL 112)
 - Optimization ANALYSIS=MFREQ or MTRAN
 - Combined modes extraction
 - Normal modes (SOL 103)
 - Case Control

$$\text{METHOD} \begin{bmatrix} \text{BOTH} \\ \text{STRUCTURE} \\ \text{FLUID} \end{bmatrix} = n$$

Equations – Fluid-Structure

- **Coupled interior acoustics**

$$\begin{bmatrix} \mathbf{M}_f & -\mathbf{A}^T \\ \mathbf{0} & \mathbf{M}_s \end{bmatrix} \begin{Bmatrix} \ddot{\mathbf{p}} \\ \ddot{\mathbf{u}} \end{Bmatrix} + \begin{bmatrix} \mathbf{B}_f & \mathbf{0} \\ \mathbf{0} & \mathbf{B}_s \end{bmatrix} \begin{Bmatrix} \dot{\mathbf{p}} \\ \dot{\mathbf{u}} \end{Bmatrix} + \begin{bmatrix} \mathbf{K}_f & \mathbf{0} \\ \mathbf{A} & \mathbf{K}_s \end{bmatrix} \begin{Bmatrix} \mathbf{p} \\ \mathbf{u} \end{Bmatrix} = \begin{Bmatrix} \mathbf{P}_f \\ \mathbf{P}_s \end{Bmatrix}$$

- **A (area) coupling matrix – struct-dof x fluid-dof**
- **$\mathbf{P}_f$ is acoustic source (ie: flow accel $\dot{Q}$) – SLOAD or ACSRCE**
- **Interior acoustic equation**

$$\mathbf{M}_f \ddot{\mathbf{p}} + \mathbf{B}_f \dot{\mathbf{p}} + \mathbf{K}_f \mathbf{p} = \mathbf{A}^T \ddot{\mathbf{u}} + \mathbf{P}_f$$

Response Types

		Response	
		Structural Displacements	Acoustic Pressure
Effect	Modes	Structural Mode	Acoustic Fluid Mode Acoustic Structural Mode
	Geometry		Acoustic Panel Acoustic Grid
	Load		Acoustic Load

Participation Factors

- **Participation factors describe the contribution of one single effect to the response considered.**
- **If the system is linear, the total response is the sum of the contributions of the single effects.**

Equations

- **PFMode(Structure) & MCFraction**

$$\mathbf{u} = \mathbf{\Phi}\mathbf{q} \rightarrow \tilde{u}_i(f_k) = \sum_j \phi_{ij} \tilde{q}_j(f_k)$$

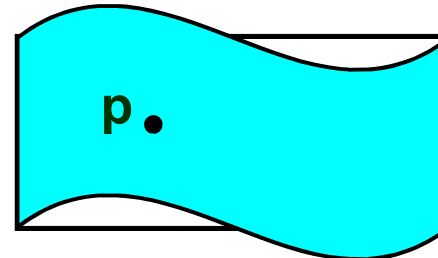
- **Each $\phi_{ij} \tilde{q}_j(f_k)$ term is the response MPF**
- **If the system is linear, the total response is the sum of the contributions of the single effects.**

Participation Factor Case Control

- **PFMODE**
 - controls computation and output of all types of modal participation factors
- **PFPANEL**
 - controls computation and output of **panel** participation factors
 - Panels define logical groups of elements
 - Ptype property ID
 - Element ID list
 - Gridpoint ID list
- **PFGRID**
 - controls computation and output of grid participation factors

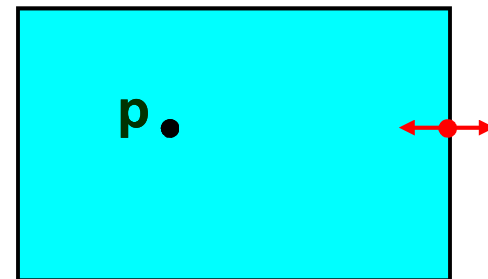
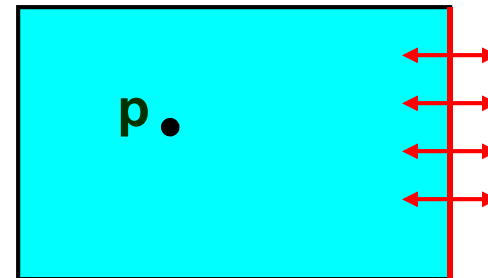
Mode Participation Factors

- **Structural Mode Participation Factors** indicate the contribution of the structural modes to the structural displacement.
- **Acoustic Fluid Mode Participation Factors** indicate the contribution of the fluid modes to the acoustic pressure.
- **Acoustic Structural Mode Participation Factors** indicate the contribution of the structural modes to the acoustic pressure.



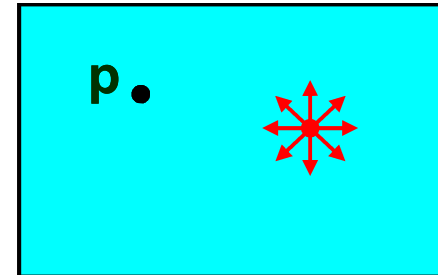
Geometric Participation Factors

- **Panel participation factors** describe the contribution of different structural regions of the fluid-structure interface to the acoustic pressure.
- **Grid participation factors** describe the contribution of the different structural grids in the fluid-structure interface to the acoustic pressure.



Load Participation Factors

- **Load Participation Factors indicate the contribution of an acoustic source to the acoustic pressure.**
- **This is the pressure due to the acoustic source in a rigid cavity.**



Mode Participation Factor Output

A C O U S T I C S T R U C T U R E M O D A L P A R T I C I P A T I O N F A C T O R S										
GRID POINT = 11217, TOTAL RESPONSE (R/I) = -1.48382E-01 / 4.54826E-02, (M/P) = 1.55196E-01 / 162.96 LOAD FREQUENCY = 3.70248E+01, (SUBCASE 1, DLOAD = 200) MAXIMUM MODAL RESP = 1.49944E-01 FOR MODE ID = 7, SORTKEY = FRACTION, SORT = ABS VALUE DESCENDING, FILTER = 1.00000E-03										
MODE	NATURAL	MODAL RESPONSE		MODAL RESPONSE		PROJECTION	REL.	MODAL		SCALED RESPONSE
ID	FREQ (HZ)	REAL	IMAGINARY	MAGNITUDE	PHASE	MAGNITUDE	PHASE	FRACTION		MAGNITUDE
7	3.72109E+01	-1.39853E-01	5.40772E-02	1.49944E-01	158.86	1.49561E-01	-4.10	9.63686E-01		9.97443E-01
2	9.75645E-05	-1.51730E-02	5.11048E-05	1.51731E-02	179.81	1.45218E-02	16.85	9.35705E-02		9.68481E-02
15	8.55370E+01	1.32441E-02	-1.87085E-03	1.33756E-02	351.96	-1.32109E-02	189.00	-8.51237E-02		-8.81054E-02
3	2.19911E-05	-1.37557E-02	5.39610E-05	1.37558E-02	179.78	1.31676E-02	16.82	8.48446E-02		8.78165E-02
14	8.21214E+01	1.01072E-02	-3.14885E-03	1.05863E-02	342.70	-1.05862E-02	179.74	-6.82117E-02		-7.06010E-02
9	6.72278E+01	-5.58743E-03	-1.82758E-03	5.87873E-03	198.11	4.80650E-03	35.15	3.09705E-02		3.20553E-02
1	1.01476E-04	4.44839E-03	6.21026E-06	4.44839E-03	0.08	-4.25125E-03	-162.88	-2.73927E-02		-2.83522E-02
5	9.13843E-05	-2.68055E-03	1.09233E-05	2.68057E-03	179.77	2.56606E-03	16.81	1.65342E-02		1.71134E-02

1

2

3

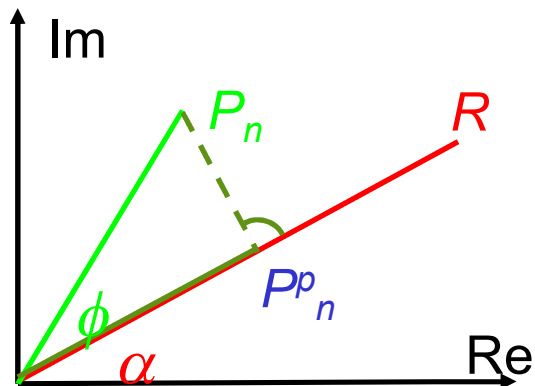
4

5

6

7

8



1	Real Part of P_n	5	P_n^p
2	Imag. Part of P_n	6	ϕ
3	Magnitude of P_n	7	$P_n^p / R $
4	Phase of P_n	8	$P_n^p / \max P_n $

Panel Participation Factor Output

ACOUSTIC PANEL PARTICIPATION FACTORS

GRID POINT = 11217, TOTAL RESPONSE (R/I) = -1.48382E-01 / 4.54826E-02, (M/P) = 1.55196E-01 / 162.96
 LOAD FREQUENCY = 3.70248E+01, (SUBCASE 1, DLOAD = 200)
 MAXIMUM PANEL RESP = 1.60744E-01 FOR PANEL = REAR, SORTKEY = FRACTION, SORT = ABS VALUE DESCENDING, FILTER = 1.00000E-03

PANEL NAME	PANEL RESPONSE		PANEL RESPONSE		PROJECTION		REL. PHASE	PANEL SCALED RESPONSE	
	REAL	IMAGINARY	MAGNITUDE	PHASE	MAGNITUDE	FRACTION		MAGNITUDE	
REAR	-1.52091E-01	5.20272E-02	1.60744E-01	161.12	1.60660E-01	-1.84	1.03521E+00	9.99483E-01	
BOTTOM	-1.28920E-01	5.65476E-05	1.28920E-01	179.97	1.23276E-01	17.02	7.94324E-01	7.66912E-01	
TOP	6.44526E-02	3.64820E-04	6.44536E-02	0.32	-6.15158E-02	-162.63	-3.96374E-01	-3.82695E-01	
FRONT	4.43994E-02	7.87686E-03	4.50927E-02	10.06	-4.01415E-02	-152.90	-2.58650E-01	-2.49724E-01	
RIGHT	1.47894E-02	-6.90774E-03	1.63231E-02	334.96	-1.61644E-02	172.01	-1.04155E-01	-1.00560E-01	
LEFT	1.47887E-02	-6.90783E-03	1.63225E-02	334.96	-1.61638E-02	172.00	-1.04151E-01	-1.00556E-01	

1

2

3

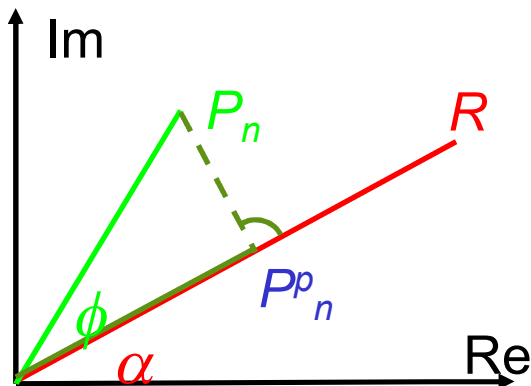
4

5

6

7

8

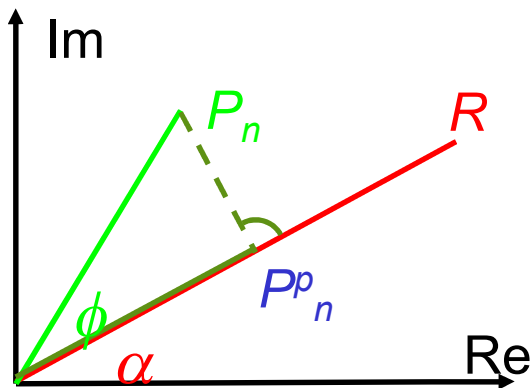


1	Real Part of P_n	5	P_n^p
2	Imag. Part of P_n	6	ϕ
3	Magnitude of P_n	7	$P_n^p / R $
4	Phase of P_n	8	$P_n^p / \max P_n $

Grid Participation Factor Output

0
 FREQUENCY = 3.707886E+01, FLUID GRID POINT = 11217
 N O R M A L I Z E D G R I D P A R T I C I P A T I O N F A C T O R S
 (REAL/IMAGINARY)

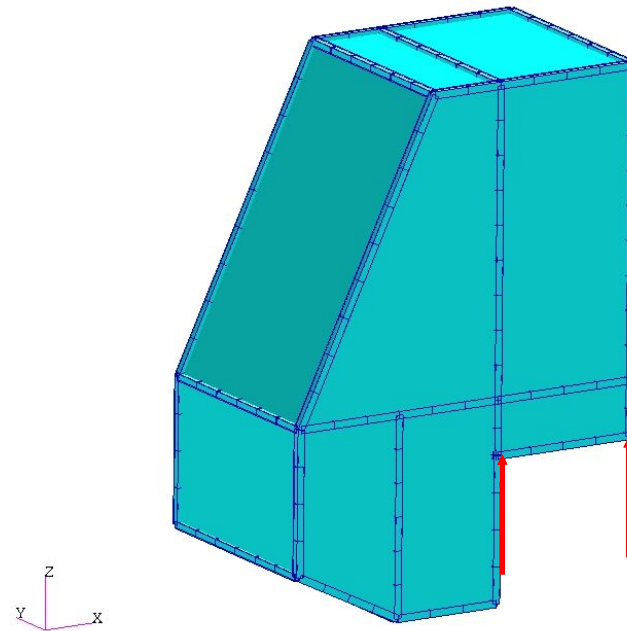
	POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
0	1	G	-1.227388E-03	-7.835577E-06	3.192602E-03	5.298424E-08	-1.137150E-06	3.031731E-08
			-1.937030E-04	-3.449941E-07	9.830291E-04	-6.015977E-09	-4.666158E-07	8.107695E-09
0	2	G	-2.451404E-03	-4.157740E-16	6.398413E-03	6.321835E-15	-3.375352E-06	-2.460994E-15
			-3.867402E-04	-3.576813E-17	1.967880E-03	-1.602187E-15	-1.208831E-06	-4.439928E-16
0	3	G	-2.449937E-03	8.598291E-24	6.398364E-03	-1.616146E-15	-4.257703E-06	-3.839641E-17
			-3.863729E-04	9.147064E-25	1.968658E-03	-7.198935E-17	-1.431066E-06	-3.576531E-17
0	4	G	-2.449655E-03	-4.416014E-27	6.399379E-03	-2.594894E-19	-4.581274E-06	-2.391980E-19
			-3.862857E-04	-1.795742E-27	1.968798E-03	-8.923359E-20	-1.512403E-06	-7.294159E-20
0	5	G	-2.449939E-03	6.102095E-24	6.398367E-03	-1.101957E-15	-4.257703E-06	-8.464838E-17
			-3.863735E-04	6.598512E-25	1.968659E-03	-4.933439E-17	-1.431066E-06	-7.821964E-17
0	6	G	-2.451408E-03	6.493559E-21	6.398419E-03	7.010387E-15	-3.375352E-06	-2.926491E-15
			-3.867414E-04	5.486240E-22	1.967883E-03	-1.774349E-15	-1.208831E-06	-5.279344E-16
0	7	G	-1.227391E-03	-7.844055E-06	3.192607E-03	5.298349E-08	-1.137149E-06	3.027685E-08
			-1.937039E-04	-3.484462E-07	9.830308E-04	-6.016375E-09	-4.666160E-07	8.095050E-09



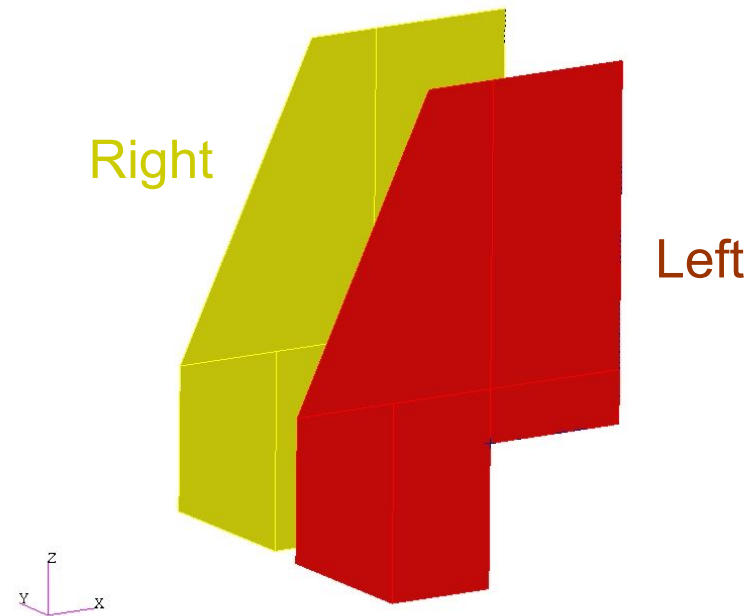
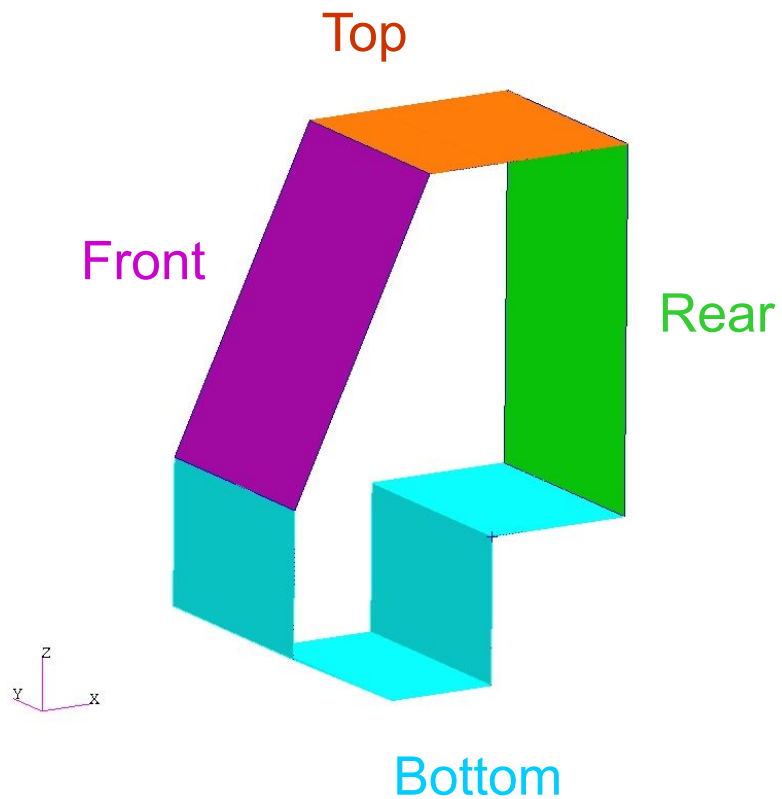
Values printed are P_n / R where P_n is the pressure at fluid grid point 11217 due to an acceleration at the corresponding degree of freedom.

Example: Problem Description

- Cabin is excited by four forces at the corners of the seat.
- Result of interest is the pressure at the driver's ear.
- Frequency response shows that 37Hz is a critical frequency.



Example: Panel Definition



Example: Input Deck

```
$ Participation Factor Test Problem: Cabin
$
$ Coupled Modal Frequency Response Analysis
$
$ Participation Factor Example - Cabin
$
$ Illustrates use of
$   o Acoustic Fluid Modal Participation Factors
$   o Acoustic Struct. Modal Part. Factors
$   o Acoustic Panel Part. Factors
$
$ =====
$
SOL 111
DIAG 8
CEND
$
ECHO=SORT (EXCEPT,CBEAM,CQUAD4,CHEXA,CPENTA,GRID)
AUTOSPC (NOZERO) = YES
$
METHOD (STRUCTURE)=1
METHOD (FLUID)    =2
$
FREQ  = 100
DLOAD = 200
$
```

**Highly recommended to ALWAYS separately
define METHODS for FLUID and STRUCTURE**

Example: Input Deck (cont.)

```
SET 1 = 11217
SET 20 = 11217
SET 90 = 37.
$
DISP(PHAS) = 1
$
PFMODE(FLUID, STRUCTMP-ALL, FLUIDMP-ALL,
      SOLUTION=90, SORT=ABSD) = 20
PFPANEL(SOLUTION=90, SORT=ABSD) = 20
$
OUTPUT(XY PLOT)
  XPAPER=29.
  YPAPER=21.
  XGRID=YES
  YGRID=YES
  XTITLE = Frequency
  YTITLE = Pressure
  XY PLOT DISP RESPONSE / 11217(T1)
$
BEGIN BULK
$
$ Request OP2 for PATRAN
PARAM, POST, -1
$
$ Define Structural Damping
PARAM, G, 0.02
$
```

Example: Input Deck (cont.)

```
$ Define Fluid Damping
PARAM, GFL, 0.002
$
$ Define Reference Pressure for dB (in Pa)
PARAM, PREFDB, 2.8284-5
$
PARAM, GRDPNT, 0          $ Request Weight Output
$
$ Structural and Acoustic Modes up to 300Hz
EIGRL, 1,,300.
EIGRL, 2,,300.
$
$ Frequency Range: 25Hz to 100Hz by Steps of 5Hz
FREQ1, 100, 25., 5., 15
$ Additional Frequencies around Resonance Frequencies
FREQ4, 100, 25., 100., 0.01, 5
$
$ Excitation Forces
RLOAD1, 200, 210,,, 220
DAREA, 210, 1, 3, 1., 7, 3, 1.
DAREA, 210, 29, 3, 1., 35, 3, 1.
TABLED1, 220, 0., 1., 1000., 1., ENDT
$
```

Example: Input Deck (cont.)

```
$ Nonmatching Fluid-Structure Interface
ACMODL, DIFF
$
$ Include Structural and Acoustic Model
INCLUDE 'cabin_structure.bdf'
INCLUDE 'cabin_fluid.bdf'
$
$ Panels
$
SET3, 101, ELEM, 127, THRU, 162, 667, THRU, 738
SET3, 201, ELEM, 37, THRU, 72, 739, THRU, 810
SET3, 301, ELEM, 331, THRU, 384
SET3, 401, ELEM, 25, THRU, 36, 73, THRU, 126
SET3, 501, ELEM, 271, THRU, 294, 601, THRU, 612
SET3, 601, ELEM, 1, THRU, 24, 455, THRU, 478,
    , 497, THRU, 562
$
PANEL, LEFT, 101, RIGHT , 201, FRONT, 301, REAR, 401
PANEL, TOP , 501, BOTTOM, 601
$
ENDDATA
```

Example: Results

A C O U S T I C S T R U C T U R E M O D A L P A R T I C I P A T I O N F A C T O R S									
GRID POINT = 11217, TOTAL RESPONSE (R/I) = -1.48382E-01 / 4.54826E-02, (M/P) = 1.55196E-01 / 162.96									
LOAD FREQUENCY = 3.70248E+01, (SUBCASE 1, DLOAD = 200)									
MAXIMUM MODAL RESP = 1.49944E-01 FOR MODE ID = 7, SORTKEY = FRACTION, SORT = ABS VALUE DESCENDING, FILTER = 1.00000E-03									
MODE ID	NATURAL FREQ (HZ)	MODAL RESPONSE		MODAL RESPONSE		PROJECTION	REL. PHASE	MODAL FRACTION	SCALED RESPONSE
		REAL	IMAGINARY	MAGNITUDE	PHASE	MAGNITUDE			MAGNITUDE
7	3.72109E+01	-1.39853E-01	5.40772E-02	1.49944E-01	158.86	1.49561E-01	-4.10	9.63686E-01	9.97443E-01
2	9.75645E-05	-1.51730E-02	5.11048E-05	1.51731E-02	179.81	1.45218E-02	16.85	9.35705E-02	9.68481E-02
15	8.55370E+01	1.32441E-02	-1.87085E-03	1.33756E-02	351.96	-1.32109E-02	189.00	-8.51237E-02	-8.81054E-02
3	2.19911E-05	-1.37557E-02	5.39610E-05	1.37558E-02	179.78	1.31676E-02	16.82	8.48446E-02	8.78165E-02
14	8.21214E+01	1.01072E-02	-3.14885E-03	1.05863E-02	342.70	-1.05862E-02	179.74	-6.82117E-02	-7.06010E-02
9	6.72278E+01	-5.58743E-03	-1.82758E-03	5.87873E-03	198.11	4.80650E-03	35.15	3.09705E-02	3.20553E-02
1	1.01476E-04	4.44839E-03	6.21026E-06	4.44839E-03	0.08	-4.25125E-03	-162.88	-2.73927E-02	-2.83522E-02
5	9.13843E-05	-2.68055E-03	1.09233E-05	2.68057E-03	179.77	2.56606E-03	16.81	1.65342E-02	1.71134E-02

- **Structural Mode #7 is responsible for the peak:**
 - Modal fraction is close to 1.
 - Relative phase is close to 0.
 - Natural frequency is close to excitation frequency.

Example: Results (cont.)

ACOUSTIC PANEL PARTICIPATION FACTORS

GRID POINT = 11217, TOTAL RESPONSE (R/I) = -1.48382E-01 / 4.54826E-02, (M/P) = 1.55196E-01 / 162.96
 LOAD FREQUENCY = 3.70248E+01, (SUBCASE 1, DLOAD = 200)
 MAXIMUM PANEL RESP = 1.60744E-01 FOR PANEL = REAR, SORTKEY = FRACTION, SORT = ABS VALUE DESCENDING, FILTER = 1.00000E-03

PANEL NAME	PANEL RESPONSE		PANEL RESPONSE		PROJECTION MAGNITUDE	REL.	PANEL	SCALED RESPONSE MAGNITUDE
	REAL	IMAGINARY	MAGNITUDE	PHASE		PHASE	FRACTION	
REAR	-1.52091E-01	5.20272E-02	1.60744E-01	161.12	1.60660E-01	-1.84	1.03521E+00	9.99483E-01
BOTTOM	-1.28920E-01	5.65476E-05	1.28920E-01	179.97	1.23276E-01	17.02	7.94324E-01	7.66912E-01
TOP	6.44526E-02	3.64820E-04	6.44536E-02	0.32	-6.15158E-02	-162.63	-3.96374E-01	-3.82695E-01
FRONT	4.43994E-02	7.87686E-03	4.50927E-02	10.06	-4.01415E-02	-152.90	-2.58650E-01	-2.49724E-01
RIGHT	1.47894E-02	-6.90774E-03	1.63231E-02	334.96	-1.61644E-02	172.01	-1.04155E-01	-1.00560E-01
LEFT	1.47887E-02	-6.90783E-03	1.63225E-02	334.96	-1.61638E-02	172.00	-1.04151E-01	-1.00556E-01

- Largest contribution is from panel REAR.
- Contribution from panels REAR and BOTTOM is in phase with response.
- Contribution of remaining panels is out of phase with response.

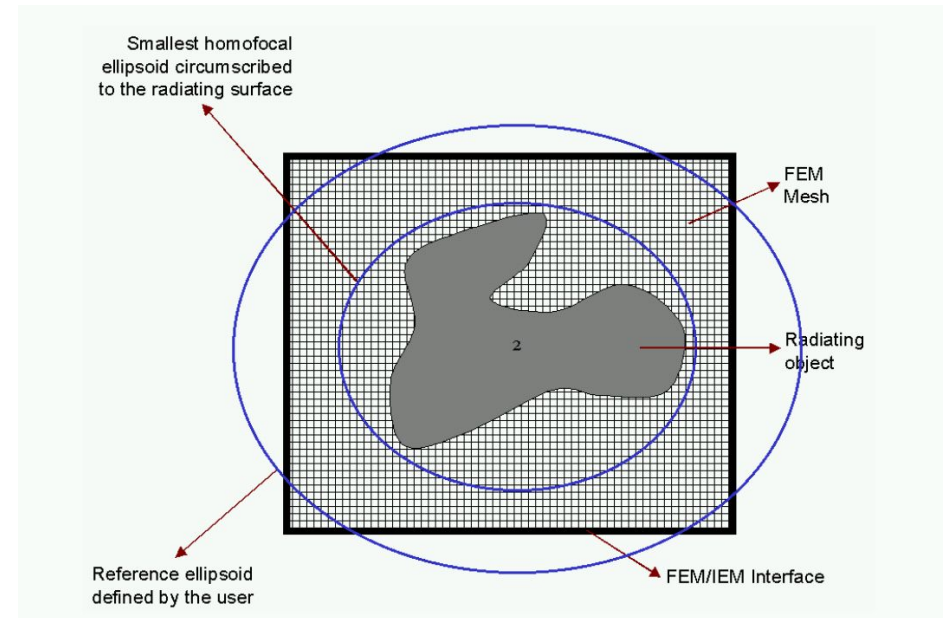
Exterior Acoustics

Objectives

- **Acoustic Analysis of Unbounded Domains**
 - Correct Non-Reflecting Boundary Condition
 - Sound Pressure in the Far Field
 - Radiated Power through Surfaces in the Far Field
 - Radiated Power from the Wetted Surface

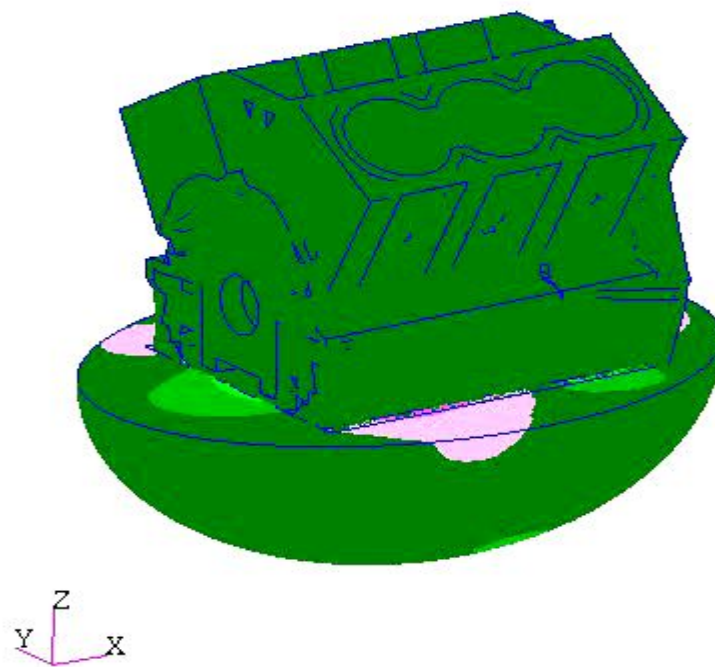
Method

- **Integration of Infinite Elements from MSC.Actran into MSC Nastran**
- **Vicinity of Radiating Body modeled using standard elements**
- **Infinite Elements provide correct boundary condition**



Applications

- Acoustic radiation from engines, oilpans, exhaust pipe systems, ...
- Sound transmission
- Not possible is point source in infinite field
- Not possible is Scattering – Sound wave hits obstical and returns
- An example is to measure the noise level for someone standing outside the car



Benefits

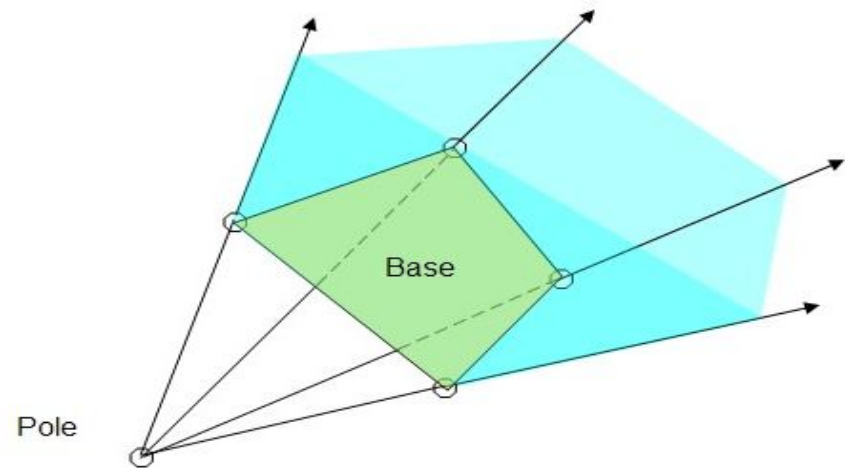
- **Proven MSC.Actran technology available within MSC Nastran**
- **Capability to model acoustic radiation, reflection and scattering**

Infinite Elements

- **Infinite Element Terminology**
- **Infinite Element Types**
- **Modeling Guidelines**
- **Computational Aspects**
- **Definition of Infinite Elements**

Infinite Element Terminology

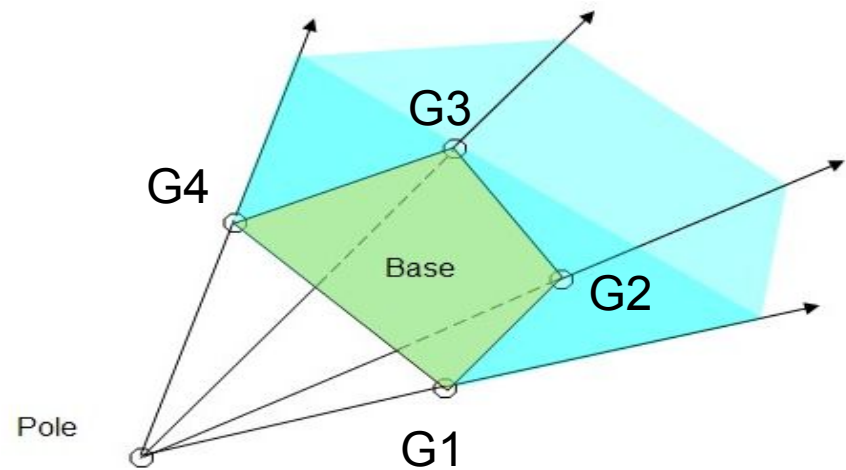
- **Geometry is described by Base and Pole.**
- **Base is attached to the finite fluid domain.**
- **Acoustic pressure is expanded into a power series of ($1/r$).**
- **Number of terms in series is called radial interpolation order.**



r = distance from pole

Infinite Element Types

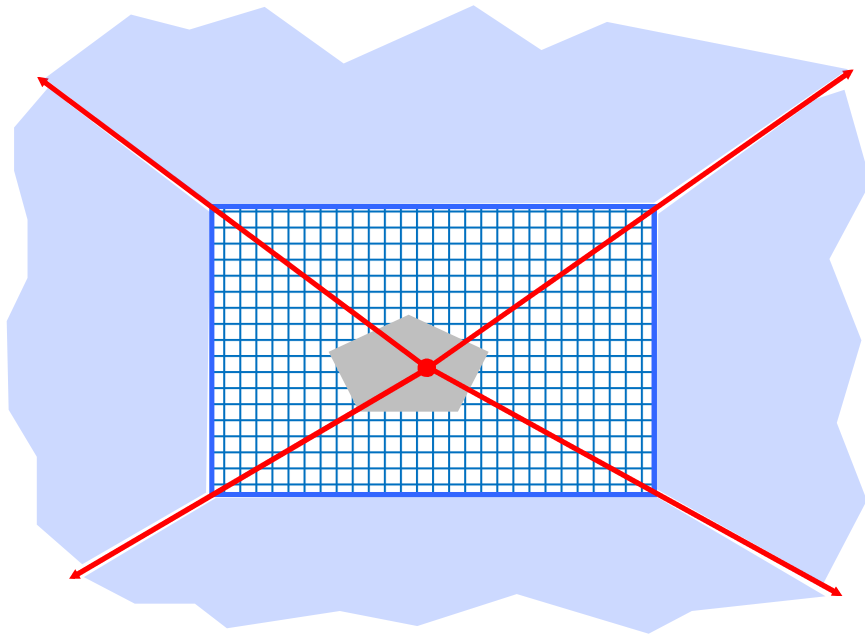
- **CACINF3** is an infinite element with a 3-noded base (similar format as CTRIA3)
- **CACINF4** is an infinite element with a 4-noded base (similar format as CQUAD4)
- **The element normal should point away from the pole.**



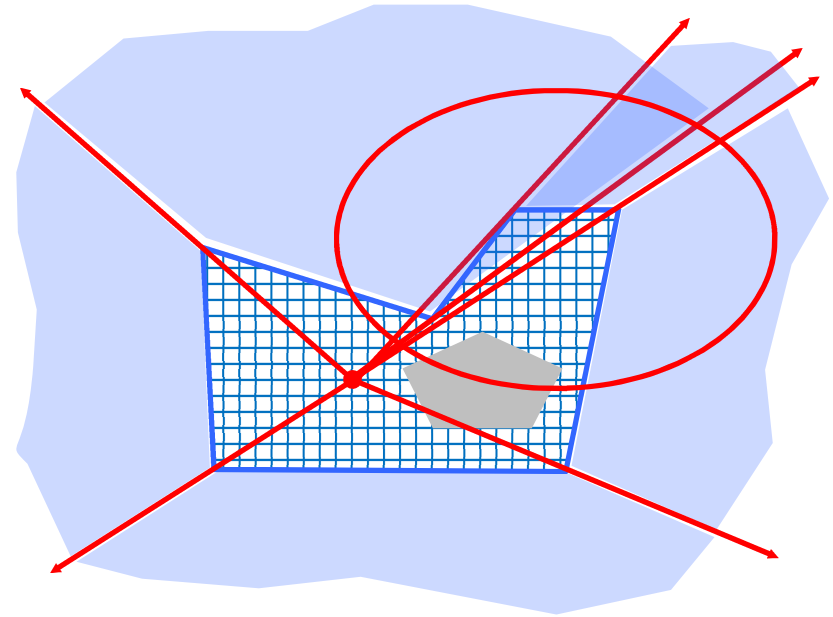
Modeling Guidelines

- **The boundary of the finite domain must be convex.**
- **The boundary of the finite domain need not be smooth.**
- **The pole must be placed such that the infinite domain is covered by the infinite elements.**

Modeling Guidelines: CONVEX BOUNDARY



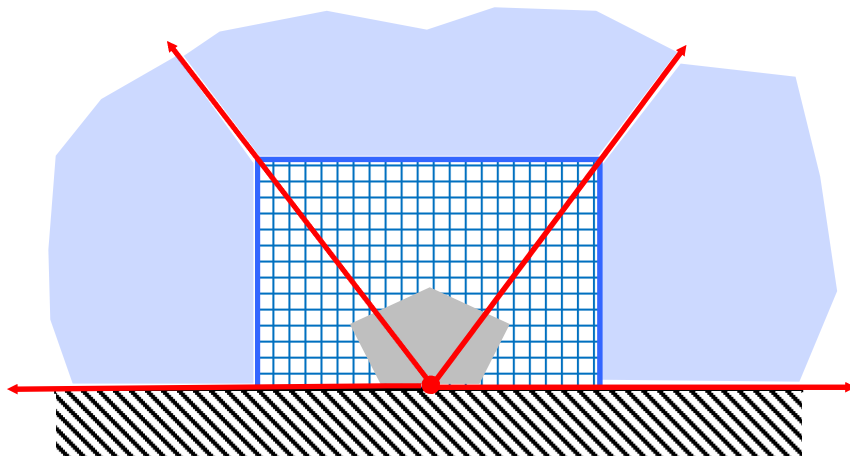
Correct



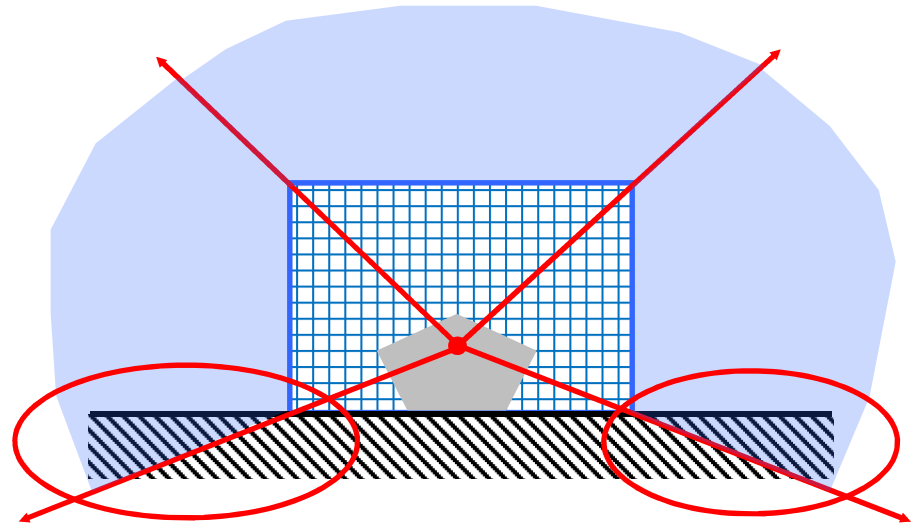
Wrong!

Modeling Guidelines: POLE LOCATION

- With halfspace problems, care has to be taken to correctly position the pole.



Correct



Wrong!

Computational Aspects

- **Infinite elements are available in the frequency domain only (SOL 108 and SOL 111).**
- **Matrices of infinite elements are unsymmetric. They are assembled into the p-set.**
- **Modal reduction of the fluid is not recommended. Modal reduction of structure is fine**

METHOD(STRUCT)=n

Definition of Infinite Elements

- **The connectivity of the infinite element base is defined on CACINF3 and CACINF4 bulk data entries.**
 - Format of CACINF3 and CACINF4 is similar to CTRIA3 and CQUAD4
- **Properties of infinite elements are defined on PACINF bulk data entries.**
- **Material properties of infinite elements are defined on MAT10 bulk data entries.**

CACINF3 and CACINF4 Entries

1	2	3	4	5	6	7	8	9	10
CACINF3	EID	PID	G1	G2	G3				

1	2	3	4	5	6	7	8	9	10
CACINF4	EID	PID	G1	G2	G3	G4			

Field	Contents
EID	Element Identification Number (Integer > 0)
PID	Property Identification Number of PACINF Entry (Integer > 0)
Gi	Grid Point Identification Numbers of Element Base Connection Points (Integer > 0)

PACINF Entry

1	2	3	4	5	6	7	8	9	10
PACINF	PID	MID	RIO	XP	YP	ZP			

Field	Contents

PID	Property Identification Number of PACINF Entry (Integer > 0)

MID	Material Identification Number of a MAT10 Entry (Integer > 0)

RIO	Radial Interpolation Order (Integer > 0)

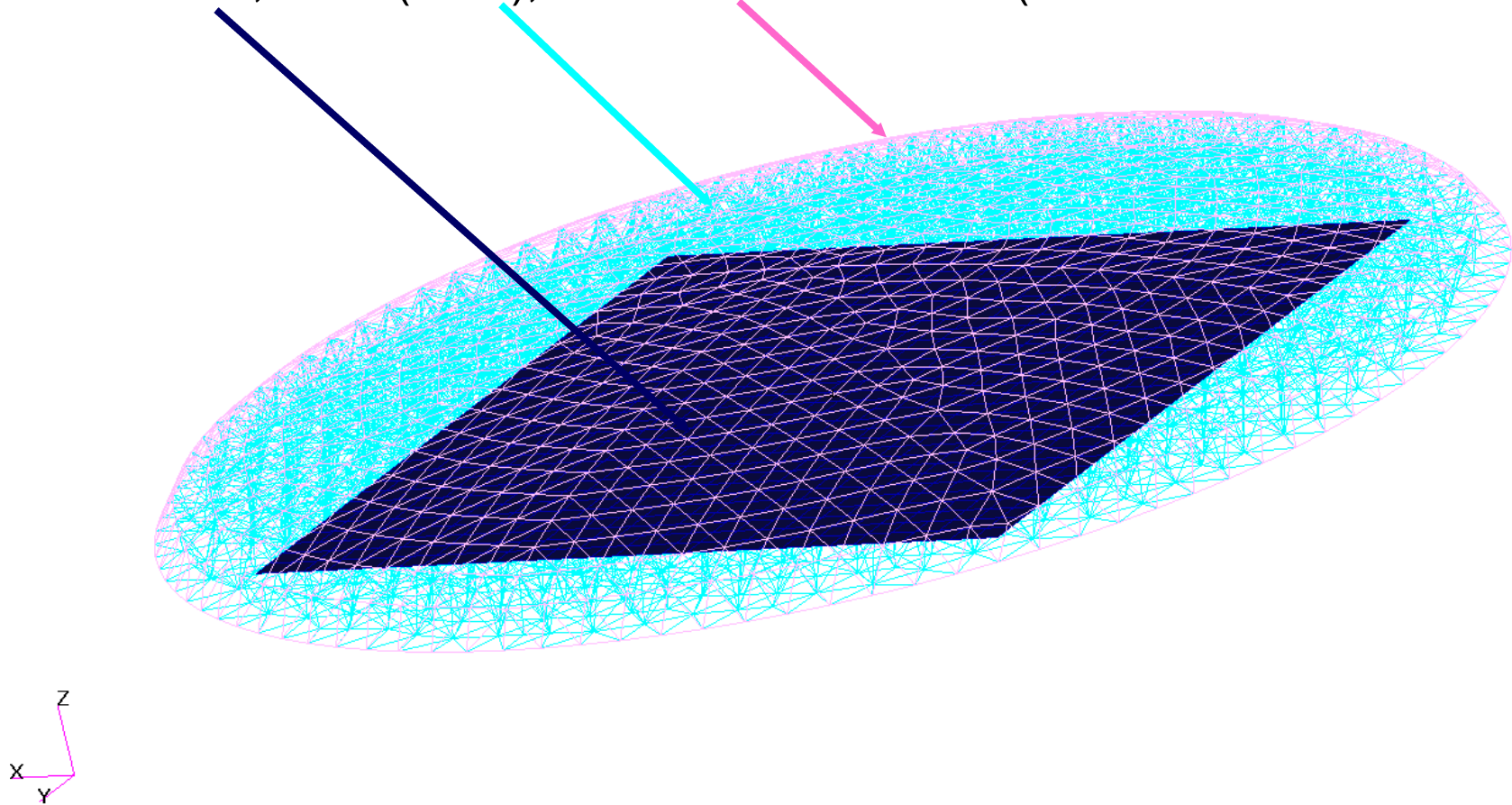
XP, YP, ZP	Coordinates of the Pole of the Infinite Elements (in the Basic Coordinate System)

Results

- **Standard acoustic results are available in the finite domain.**
- **Additional results are available at field points inside the infinite domain.**

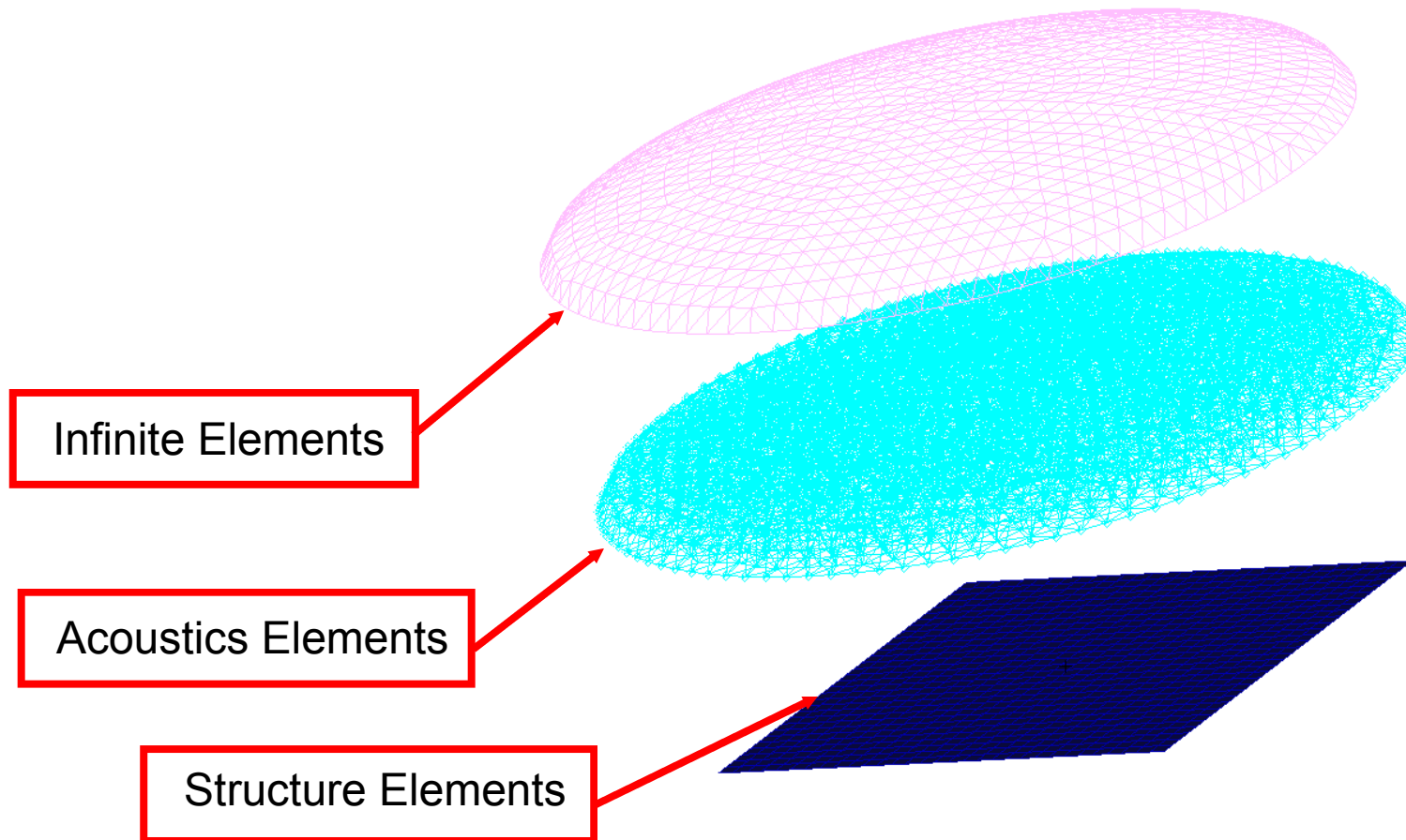
Example

- Structure, Fluid (solid), and Infinite Elements (CACINF3 or CACINF4)



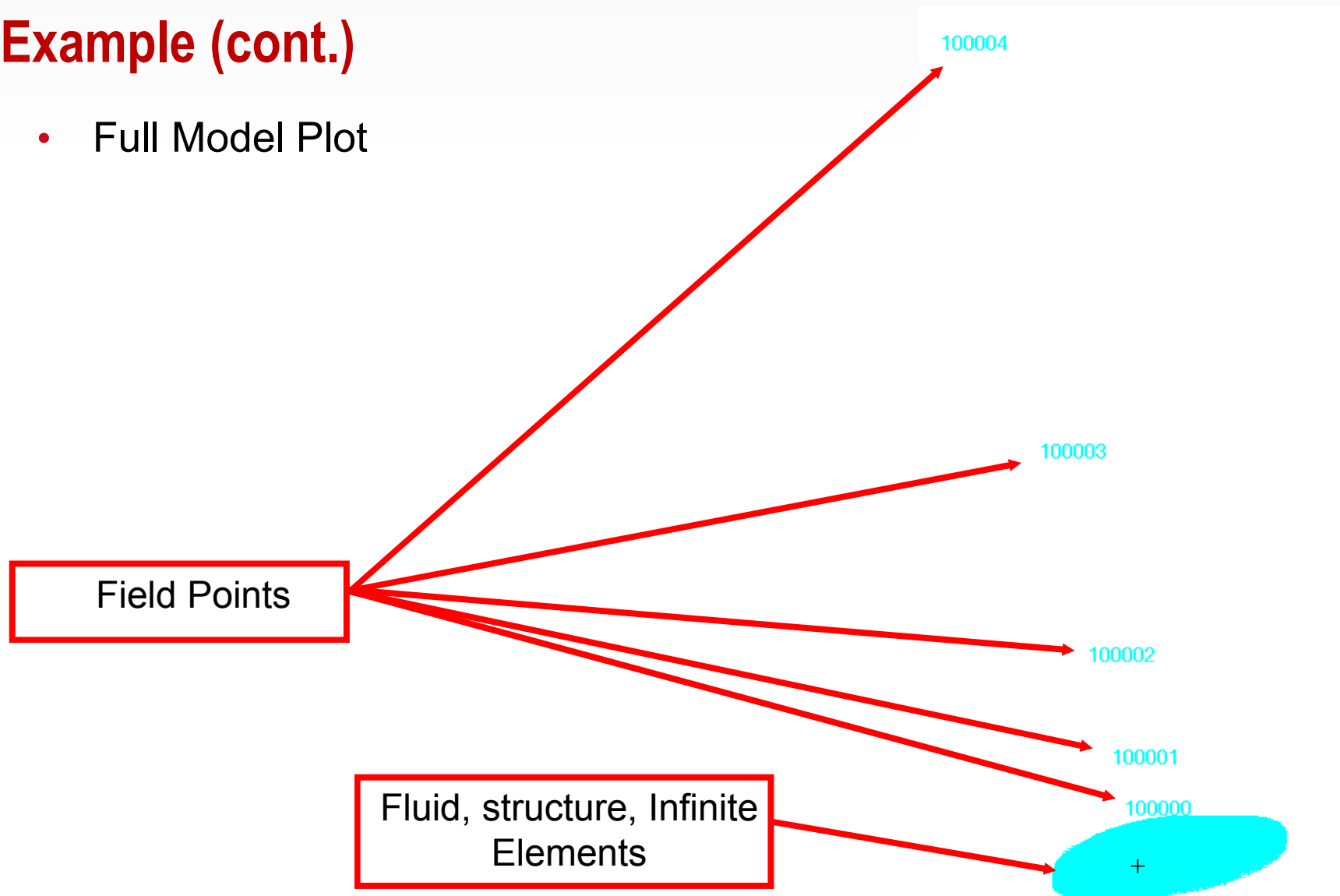
Example

- **Exploded View**



Example (cont.)

- Full Model Plot



Input File

• Typical Structure Input

```
$ Elements and Element Properties for region
PSHELL 1 1 .01 1 1
$ Pset: "plaque" will be imported as: "pshell.1"
CQUAD4 1 1 10539 33 34 2
CQUAD4 2 1 2 3 35 34
CQUAD4 3 1 3 4 36 35
.
.
```

■ Typical Infinite Elements

```
$ Elements and Element Properties for region :
PACINF 3 2 7 0. 0. 0.
$ACIFPRM EPTINF 1 PRINT 1
$ Pset: "infini" will be imported as: "pshell.3"
CACINF3 100000 3 6993 6994 6992
CACINF3 100001 3 6995 6992 6994
CACINF3 100002 3 6994 7003 6995
```

• Typical Acoustics Input

```
CTETRA 6214 2 7265 7364 7359 7131
CTETRA 6215 2 7458 7122 7130 7137
CTETRA 6216 2 7390 7148 7137 7153
CTETRA 6217 2 7410 7641 7642 7639
.
PSOLID 2 2 0 PFLUID
MAT10 2 1.2 340.
.
GRID 8525 .0850594.44994 0. -1
GRID 8526 .16094 .39993 0. -1
GRID 8527 .135964 .44994 0. -1
GRID 8528 .110242 .49995 0. -1
GRID 8529 .0850293.549914 0. -1
GRID 8530 .16094 .49995 0. -1
GRID 8531 .13583 .549878 0. -1
GRID 8532 .0581079.599815 0. -1
```

■ Acoustic Field Points

```
BEGIN AFPM=1
GRID,100000,,0.,0.,0.5
GRID,100001,,0.,0.,1.
GRID,100002,,0.,0.,2.
GRID,100003,,0.,0.,4.
GRID,100004,,0.,0.,8.
ENDDATA
```

Field Point Results

- **The following results are available at field points:**
 - Acoustic pressure
 - Acoustic velocity
 - Acoustic intensity
- **If a field point mesh is defined, the following additional results are available:**
 - Intensity component normal to field point mesh
 - Acoustic power through the field point mesh

Definition of Field Point Meshes

- Field point meshes are defined in separate sections of the bulk data file.
- These sections follow the main bulk data section.
- Each of the sections begins with

or `BEGIN BULK AFPM = afpmid`

`BEGIN AFPM = afpmid`

- where *afpmid* is the acoustic field point mesh identifier (Integer > 0).

Field Point Mesh Bulk Data Entries

- **Field points are defined using standard GRID bulk data entries.**
- **It is not necessary to place a “-1” in field 7.**
- **Field points can be connected by any type of elements.**
- **Only CQUAD4 and CTRIA3 elements define a field point mesh.**
- **PSHELL identifiers have to be specified (field 3 of the CQUAD4, CTRIA3) but the referenced PSHELL entries need not be defined.**

Field Point Mesh Output Parameters

- If output to an .op2 file is desired, **PARAM,POST,-1** must be defined within the acoustic field point mesh section.
- Output of different field point meshes can be sent to different .op2 files if parameter **OUNIT2** is defined within the acoustic field point mesh section.

Field Point Mesh Example

```
BEGIN AFPM=100
```

```
$  
PARAM, POST, -1  
$  
$ Isolated Field Points along a Line  
$  
GRID      1001      2      5.      0.      0.  
GRID      1002      2      6.      0.      0.  
GRID      1003      2      7.      0.      0.  
GRID      1004      2      8.      0.      0.  
GRID      1005      2      9.      0.      0.  
GRID      1006      2     10.      0.      0.  
CORD2R,    2,, 0., 0., 0., -.5, -.5, .707107  
          , .5, .5, .707107
```

```
$
```

```
BEGIN AFPM = 200
```

```
$  
PARAM, POST, -1  
$  
$ Field Point Mesh  
$  
GRID, 1,, -1., -1., 2.  
GRID, 2,, 0., -1., 2.  
GRID, 3,, 1., -1., 2.  
GRID, 4,, -1., 0., 2.  
GRID, 5,, 0., 0., 2.
```

Field Point Mesh Example (cont.)

```
GRID, 6,, 1., 0., 2.
GRID, 7,, -1., 1., 2.
GRID, 8,, 0., 1., 2.
GRID, 9,, 1., 1., 2.
GRID, 11,, -1., -1., 2.
GRID, 12,, 0., -1., 3.
GRID, 13,, 1., -1., 2.
GRID, 14,, -1., 0., 3.
GRID, 16,, 1., 0., 3.
GRID, 17,, -1., 1., 2.
GRID, 18,, 0., 1., 3.
GRID, 19,, 1., 1., 2.
$
CQUAD4, 1, 1, 1, 2, 5, 4
CQUAD4, 2, 1, 2, 3, 6, 5
CQUAD4, 3, 1, 4, 5, 8, 7
CQUAD4, 4, 1, 5, 6, 9, 8
$
CQUAD4, 5, 1, 12, 16, 18, 14
CTRIA3, 6, 1, 11, 12, 14
CTRIA3, 7, 1, 12, 13, 16
CTRIA3, 8, 1, 16, 19, 18
CTRIA3, 9, 1, 14, 18, 17
$
ENDDATA
```

Case Control Commands

- The **ACFPMRESULT** command controls output of acoustic field point mesh results.
- The **ACPOWER** command controls output of the power radiated from the wetted surface.
- The **INTENSITY** command controls output of acoustic intensity on the wetted surface.

Example: Input Deck – Using UMFPACK

```
NASTRAN SYSTEM(209)=16
```

UMFPACK Sparse Solver

```
SOL 108
CEND
TITLE      = Vibrating Plate Example
SUBTITLE   = Direct Frequency Response
$
ECHO = SORT(EXCEPT, GRID, CHEXA, CQUAD4, CACINF4)
$
DLOAD      = 10
FREQ       = 20
SPC        = 1
$
DISP(PLOT) = ALL
ACFPMRESULT(PHAS) = ALL
$
BEGIN BULK
$
PARAM, POST, -1
ACMODL, IDENT
$
RLOAD1, 10, 200,,, 300
PLOAD2, 200, 1., 1, THRU, 150
TABLED1, 300
          , 0., 1., 1000., 1., ENDT
$
FREQ, 20, 10.
$
```

Field Point Mesh Results for
all Field Point Meshes

Example: Input Deck (cont.)

```
$  
$ Fluid  
GRDSET,,,,, -1  
INCLUDE 'fluid1.bdf'  
$  
$ Structure  
INCLUDE 'structure.bdf'  
$  
BEGIN AFPM=100  
$  
$ Isolated Field Points  
GRID, 10001,, 0., 0., 3.5  
GRID, 10002,, 0., 0., 5.  
GRID, 10003,, 0., 0., 10.  
$  
BEGIN AFPM=200  
$  
$ Field Point Mesh  
INCLUDE 'fpm.bdf'  
$  
ENDDATA
```

Example: Input Deck – Using ITER SOLV

```
SOL 108
CEND
TITLE      = Vibrating Plate Example
SUBTITLE   = Direct Frequency Response
$
ECHO = SORT(EXCEPT, GRID, CHEXA, CQUAD4, CACINF4)
$
DLOAD      = 10
FREQ       = 20
SMETHOD   = 30
SPC        = 1
$
DISP(PLOT) = ALL
ACFPMRESULT(PHAS) = ALL
$
BEGIN BULK
$
PARAM, POST, -1
ACMODL, IDENT
$
RLOAD1, 10, 200,, 300
PLOAD2, 200, 1., 1, THRU, 150
TABLED1, 300
          , 0., 1., 1000., 1., ENDT
$
FREQ, 20, 10.
$
```

Parameters of Iterative Solver

Field Point Mesh Results for
all Field Point Meshes

Example: Input Deck (cont.)

```
$ Parameters of Iterative Solver
```

```
ITER, 30
```

```
PRECOND = J,
```

```
ITSEPS = 1.0E-4
```

Parameters of Iterative Solver

```
$
```

```
$ Fluid
```

```
GRDSET,,,,, -1
```

```
INCLUDE 'fluid1.bdf'
```

```
$
```

```
$ Structure
```

```
INCLUDE 'structure.bdf'
```

```
$
```

```
BEGIN AFPM=100
```

```
$
```

```
$ Isolated Field Points
```

```
GRID, 10001,, 0., 0., 3.5
```

```
GRID, 10002,, 0., 0., 5.
```

```
GRID, 10003,, 0., 0., 10.
```

```
$
```

```
BEGIN AFPM=200
```

```
$
```

```
$ Field Point Mesh
```

```
INCLUDE 'fpm.bdf'
```

```
$
```

```
ENDDATA
```

Example: Part of Fluid Mesh Definition

```

$ Exterior Acoustics - Vibrating Plate Example
$ Fluid Model of Domain 1: Finite and Infinite Elements
$
$ -----
$
$      PID      MID      RIO      XP      YP      ZP
PACINF  10       20       10       0.       0.       0.
PSOLID  2        20
MAT10   20       1.21    340.
$
CHEXA   1001     2       1001    1002    1018    1017    1177    1178
        1194     1193
CHEXA   1002     2       1002    1003    1019    1018    1178    1179
        1195     1194
...
$ Infinite Elements
$
CACINF4 13901    10       3641    3642    3658    3657
CACINF4 13902    10       3642    3643    3659    3658
CACINF4 13903    10       3643    3644    3660    3659
CACINF4 13904    10       3644    3645    3661    3660
CACINF4 13905    10       3645    3646    3662    3661
CACINF4 13906    10       3646    3647    3663    3662
...

```

Example: Field Point Mesh Results

0

FREQUENCY = 1.000000E+01

ACOUSTIC FIELD POINT MESH = 100

ACOUSTIC FIELD POINT MESH RESULTS

POINT ID.	ACOUSTIC PRESSURE MAGNITUDE	ACOUSTIC PRESSURE PHASE	INTENSITY COMP. NORMAL TO FPM	INTENSITY X	INTENSITY Y	INTENSITY Z
10001	5.867592E-03	3.224626E+02	0.0	3.901852E-16	1.102822E-09	4.101243E-08
10002	4.148942E-03	3.068220E+02	0.0	1.168128E-16	3.946748E-10	2.071128E-08
10003	2.089879E-03	2.542063E+02	0.0	1.290767E-17	5.147245E-11	5.294778E-09

1 VIBRATING PLATE EXAMPLE

DECEMBER 6, 2005 MSC.NASTRAN 12/ 5/05 PAGE 16

DIRECT FREQUENCY RESPONSE

0

*** USER INFORMATION MESSAGE 3119 (AFPINI)

DATA RECOVERY OF ACOUSTIC FIELD POINT MESH 200 INITIATED

1 VIBRATING PLATE EXAMPLE

DIRECT FREQUENCY RESPONSE

0

FREQUENCY = 1.000000E+01

ACOUSTIC FIELD POINT MESH = 200

ACOUSTIC FIELD POINT MESH RESULTS

POINT ID.	ACOUSTIC PRESSURE MAGNITUDE	ACOUSTIC PRESSURE PHASE	INTENSITY COMP. NORMAL TO FPM	INTENSITY X	INTENSITY Y	INTENSITY Z
46567	2.959861E-03	2.856447E+02	7.442431E-09	-7.647572E-09	7.442431E-09	1.545178E-10
46568	2.930327E-03	2.848879E+02	7.258169E-09	-7.408595E-09	7.258169E-09	1.340973E-09
46569	2.846820E-03	2.826632E+02	6.691689E-09	-6.762688E-09	6.691689E-09	2.608988E-09
46570	2.722373E-03	2.790949E+02	5.876647E-09	-5.877888E-09	5.876647E-09	3.492858E-09
46571	2.572942E-03	2.743544E+02	4.970530E-09	-4.929979E-09	4.970530E-09	3.970974E-09
46572	2.412938E-03	2.686276E+02	4.189022E-09	-4.042182E-09	4.189022E-09	4.023463E-09
46573	3.274509E-03	2.928127E+02	1.022698E-08	-8.121219E-09	1.022698E-08	2.107736E-10
46574	3.234508E-03	2.919741E+02	9.893179E-09	-7.820792E-09	9.893179E-09	1.809509E-09
46575	3.122908E-03	2.895207E+02	8.920301E-09	-7.024057E-09	8.920301E-09	3.457678E-09
46576	2.960447E-03	2.856181E+02	7.592750E-09	-5.968274E-09	7.592750E-09	4.507941E-09
46577	2.770893E-03	2.804857E+02	6.203833E-09	-4.882053E-09	6.203833E-09	4.973865E-09
46578	2.573859E-03	2.743491E+02	4.944810E-09	-3.905830E-09	4.944810E-09	5.004789E-09
46579	3.597602E-03	2.988595E+02	1.353951E-08	-8.067808E-09	1.353951E-08	2.802701E-10
46580	3.544644E-03	2.979372E+02	1.298966E-08	-7.718223E-09	1.298966E-08	2.386323E-09
46581	3.399051E-03	2.952536E+02	1.145273E-08	-6.810243E-09	1.145273E-08	4.470247E-09
46582	3.192353E-03	2.910229E+02	9.454744E-09	-5.649165E-09	9.454744E-09	5.666677E-09
46583	2.958238E-03	2.855169E+02	7.476965E-09	-4.503807E-09	7.476965E-09	6.064790E-09
46584	2.721970E-03	2.790005E+02	5.885413E-09	-3.516635E-09	5.885413E-09	5.822169E-09

Isolated Field Points, No Normal Defined

Isolated Field Points, No Normal Defined

Guidelines

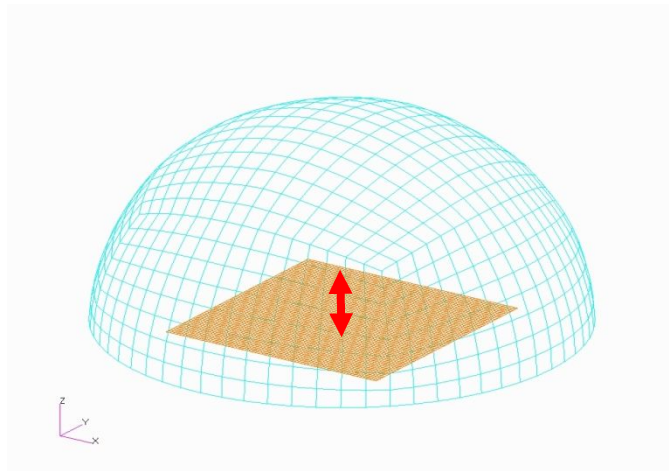
- **The surface the infinite elements are attached to must be convex but need not be smooth.**
- **The radial interpolation order required depends on the directivity of the pressure field.**
- **It is recommended to study the sensitivity of the results with respect to the radial interpolation order.**

Guidelines (cont.)

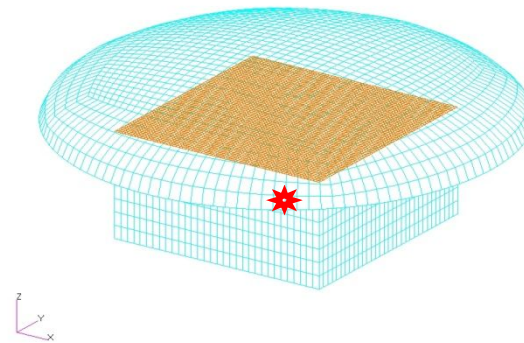
- **Infinite elements are available in SOL 108 and SOL 111.**
- **Modal reduction can be applied to the structure.**
- **Modal reduction of the fluid is not recommended.**
- **Use of the iterative solver might increase efficiency.**
- **For small-medium size UMFPACK sparse solver gives excellent efficiency over default solver.**

Accuracy Validation Test

- Compare the results of MSC Nastran external acoustic with those from MSC Actran
- Two test models are used
 - Model 1
 - Structure and exterior acoustic
 - Model 2
 - Structure, exterior and interior acoustic



Model 1

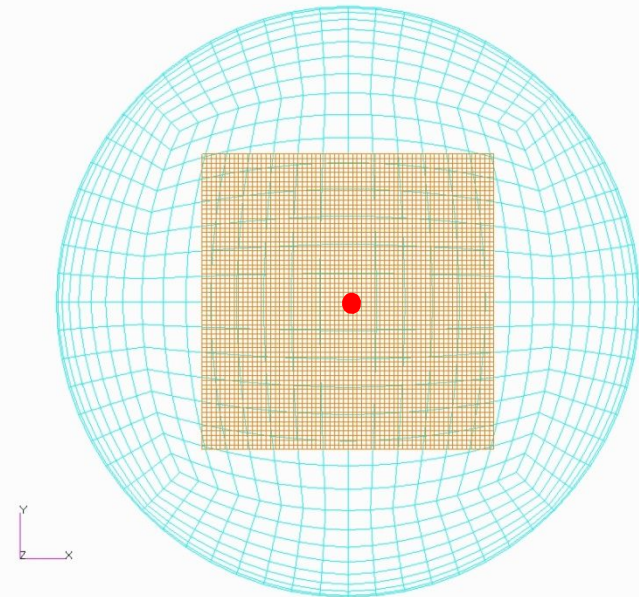
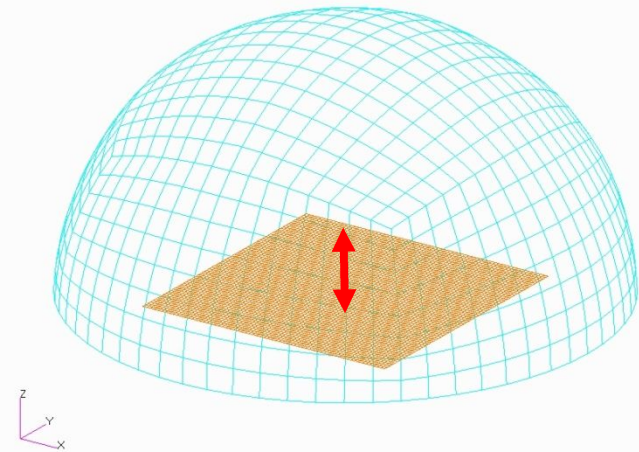


Model 2

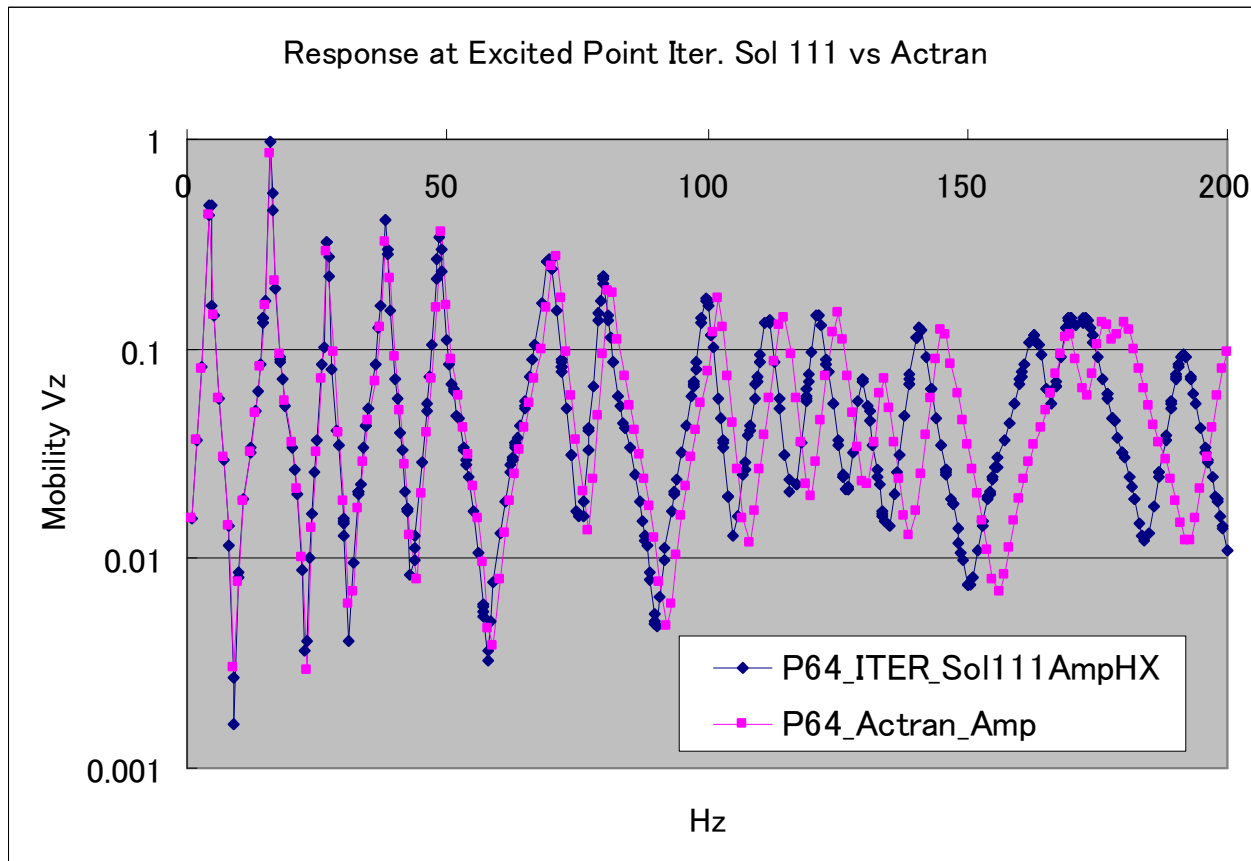
Model 1 (Exterior)

- **Unit Point Load**
- **MSC Nastran**
 - SOL108/111
- **Actran**
 - Krylov solver
- **Number of Mesh**

Plate64	Elm
air	6656
infinite	768
plate	4096
Total	11520



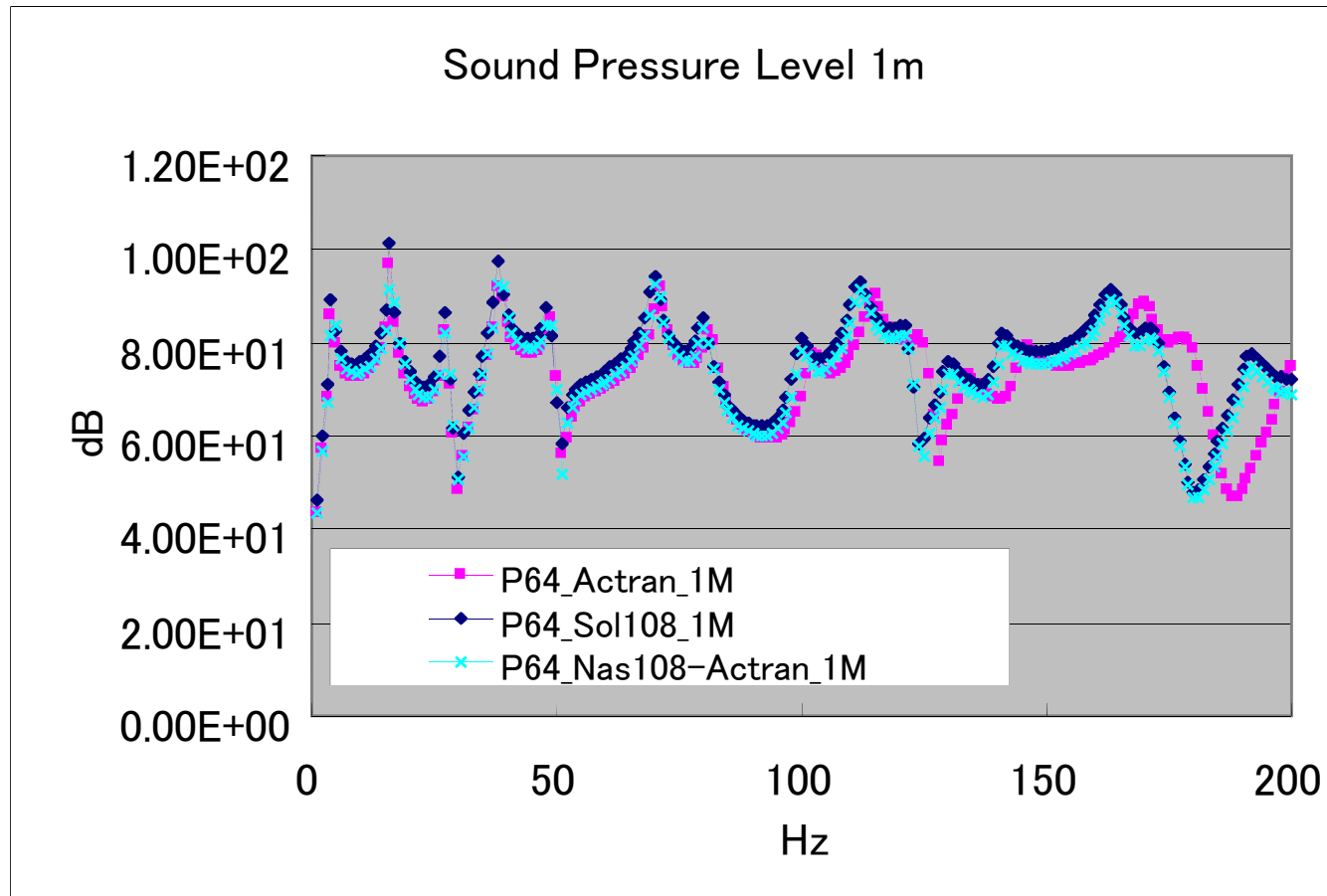
Results: FRF (Velocity) of the Center Point



In high freq. range, Actran gave “STIFF” results due to using solid shell.

Comparison of Results (Coupled vs. Decoupled)

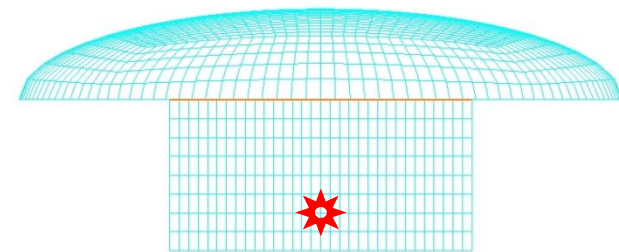
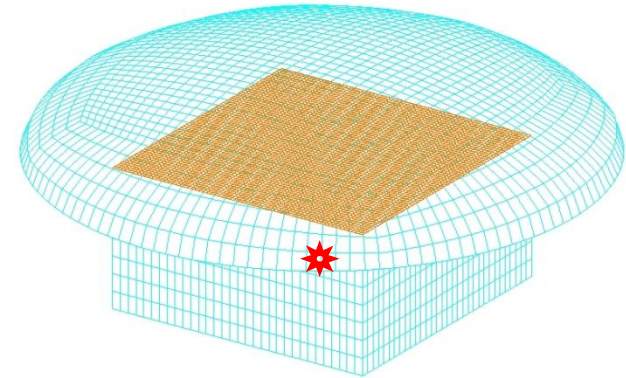
Decoupled analysis: Nastran SOL 108 structure resp. -> Actran radiation acoustics



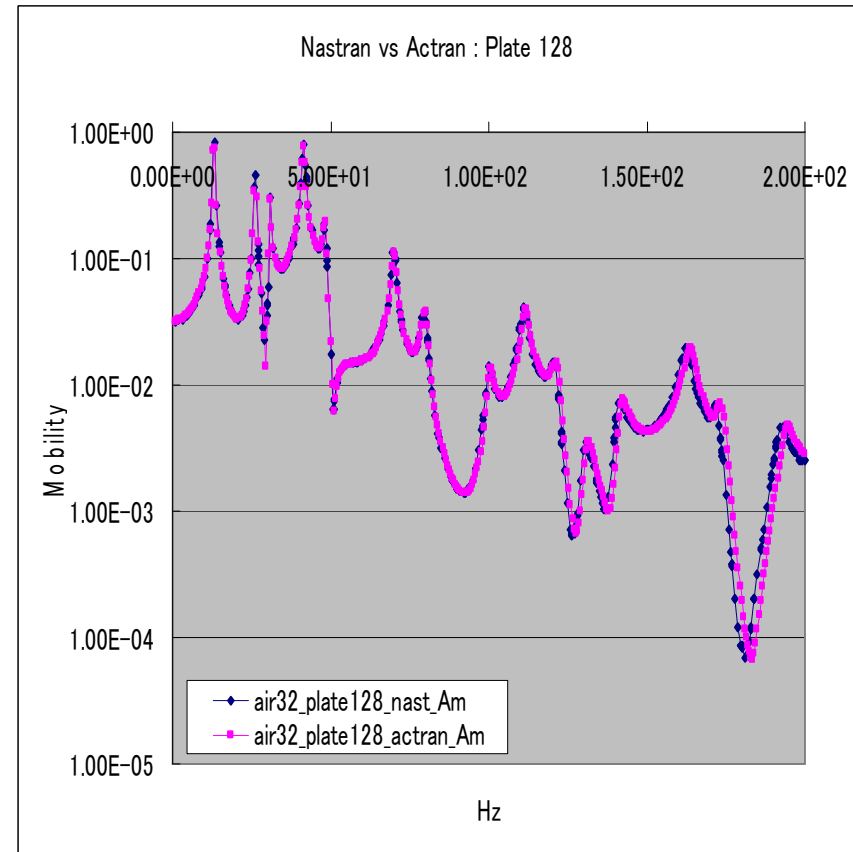
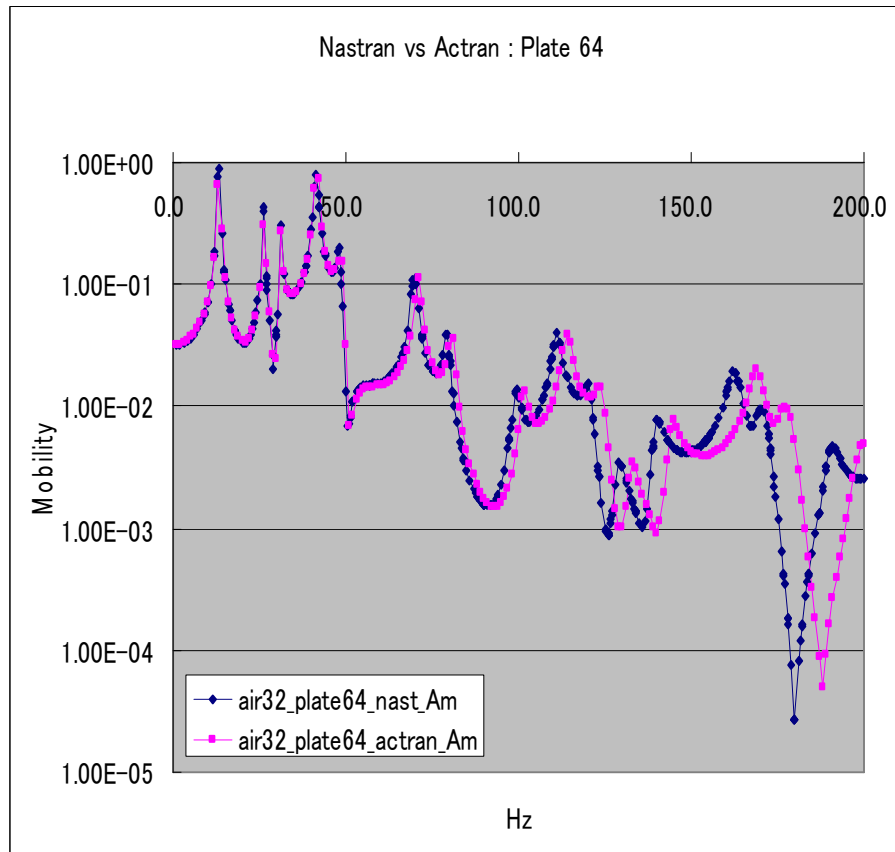
Model 2 (Interior + Exterior)

- **Acoustic Source**
 - Source Strength:0.01
- **MSC Nastran**
 - SOL111, Iterative
- **Actran**
 - Krylov solver
- **Number of Mesh**

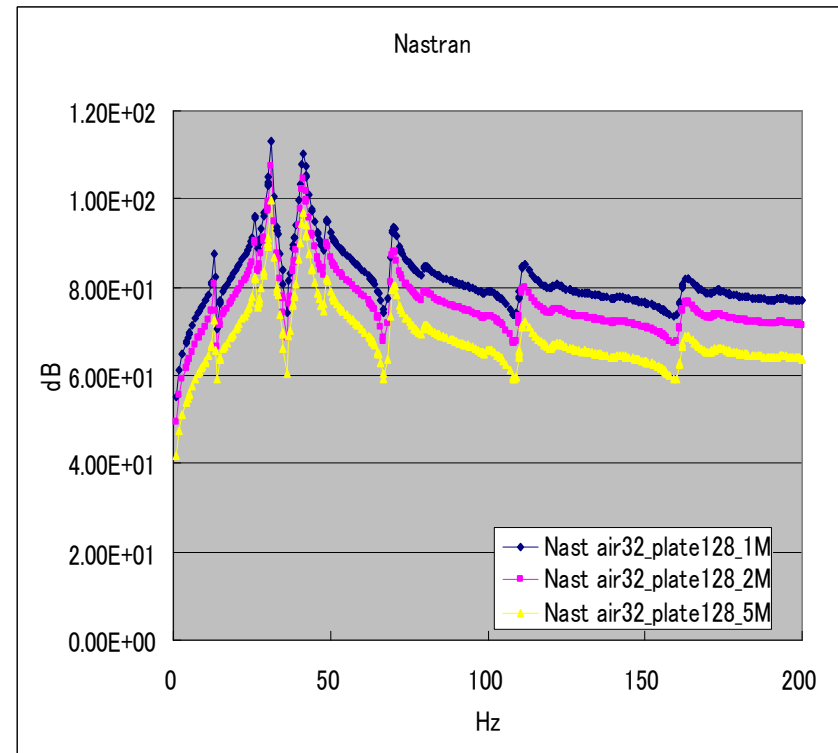
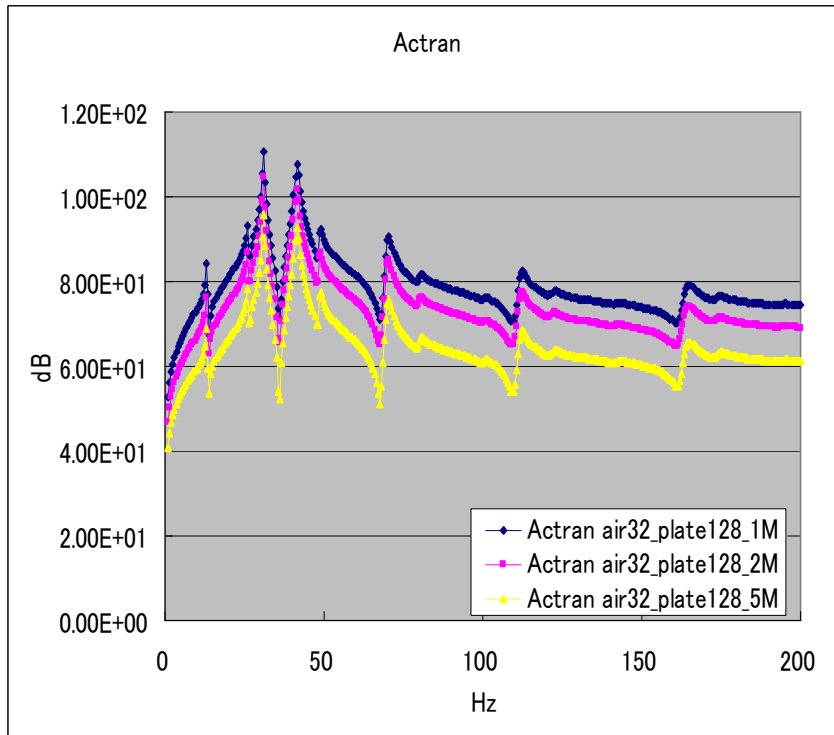
	Plate64	Plate128
air	13440	13440
infinite	1664	1664
air_low	8192	8192
plate	4096	16384
Total	27392	39680



Results: FRF of the Center Point



Results: Sound Pressure Level :(plate 128)



Remarks on Exterior Acoustics

- **Results from MSC Nastran agree with those from Actran**
- **Combined interior/exterior acoustic problem can be solved together**
- **Can combine MSC Nastran powerful solver as well as advanced MSCACMS/DMP to reduce the computational time of solving very large scale interior/exterior acoustic problems**

SECTION 15

Dynamic Design Optimization

What is “Design Optimization”?

- **Automated modifications of the analysis model parameters to achieve a desired objective while satisfying specified design requirements.**
- **WHAT ARE THE POSSIBLE APPLICATIONS?**
 - Structural design improvements (optimization)
 - Generation of feasible designs from infeasible designs
 - Model matching to produce similar structural responses
 - System parameter identification
 - Configuration evaluations
 - Correlates analytical model with test results (see Section 8)
 - Others - (depends on designer's creativity)

Basic Features Implemented in MSC NASTRAN

- **Easy access to design synthesis capabilities**
 - Concept of design model
- **Flexibility for design model representation**
 - User-supplied equation interpretation capability
- **Efficient solution for problems of “any” size**
 - Number of finite element analyses as the measure of efficiency

Strengths of MSC NASTRAN Structural Optimization

- **Efficient performance for small- to large-scale problems**
- **Reliable convergence characteristics**
- **Flexible user interface and user-defined equations**
- **Full implementation of approximation concepts**
- **Continuous enhancements**
- **Results dependent on the proven reliability of MSC Nastran analysis**
- **High-level support as a part of MSC Nastran**
- **Access to the familiar analysis tools in MSC Nastran**

Design Optimization in MSC Nastran

- **The ANALYSIS Case Control request allows you to specify the type of optimization analysis discipline that you want to perform for each subcase.**
- **Multi-Disciplinary**
- **The following analysis types are allowed in the ANALYSIS Case Control request:**
 - STATICS Statics
 - MODES Normal Modes
 - BUCK Buckling
 - DFREQ Direct Frequency
 - MFREQ Modal Frequency
 - MTRAN Modal Transient
 - SAERO Static Aeroelasticity
 - FLUTTER Flutter

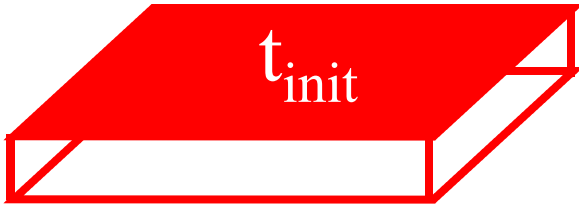
Optimization Capabilities Supported

- **Multi-Disciplinary Optimization**
- **Static Response Optimization**
- **Buckling Response Optimization**
- **Dynamic Response Optimization**
 - Direct Frequency
 - Modal Frequency
 - Modal Transient
 - Acoustics
- **Superelement Optimization**
 - Allows the design model to span superelement boundaries.
- **Aeroelastic Optimization**
 - Static Aeroelastic
 - Flutter

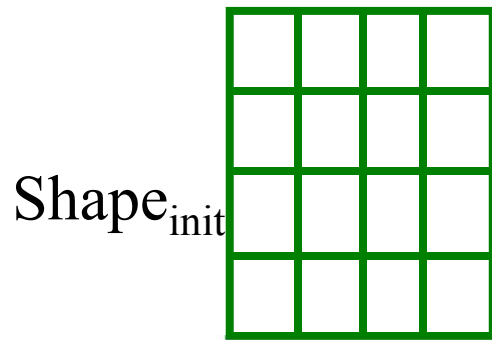
Optimization Capabilities Supported (cont.)

- **Shape Optimization**
- **Four methods are available for Basis Vector Generation**
 - Manual Grid Variation
 - Direct Input of Shapes
 - Geometric Boundary Shapes
 - Analytic Boundary Shapes

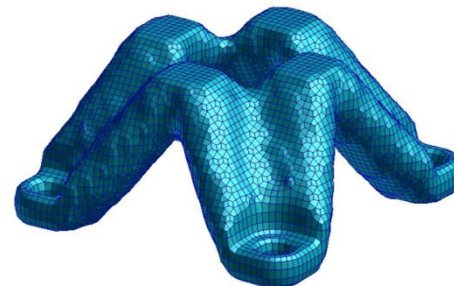
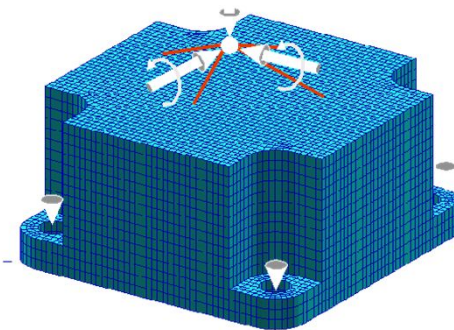
Types of Optimization in MSC Nastran



- Sizing
 - Change in property (t, I, J, theta, etc.)



- Shape
 - Change in GRID locations



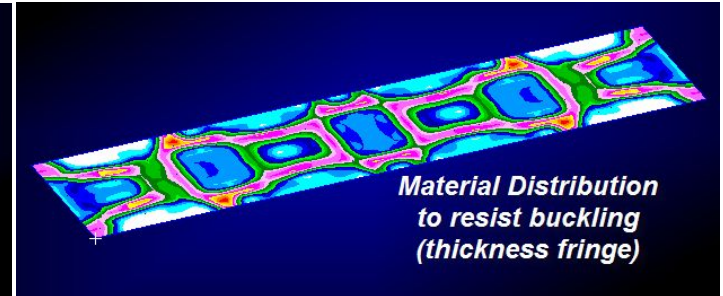
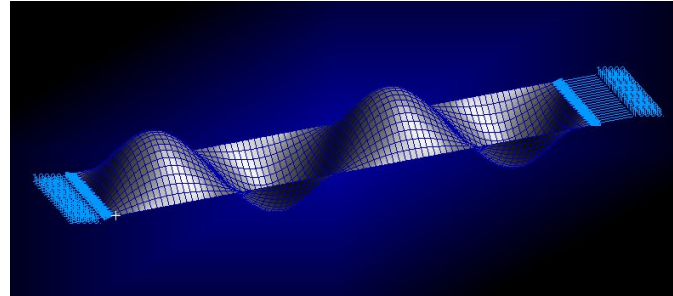
- Topology
 - “Remove” unneeded structure

$$\begin{aligned}\rho &= \rho_o \times \\ E &= E_o \times p\end{aligned}$$

Types of Optimization in MSC Nastran (cont.)

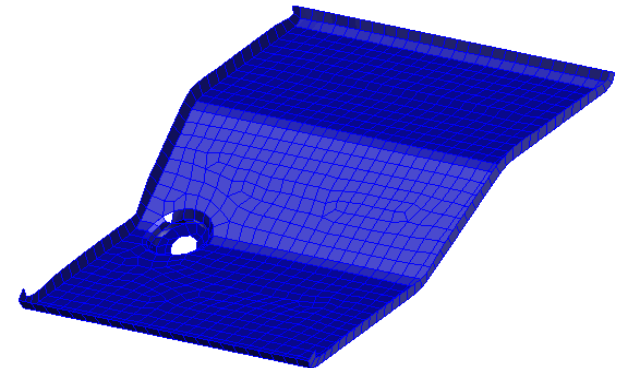
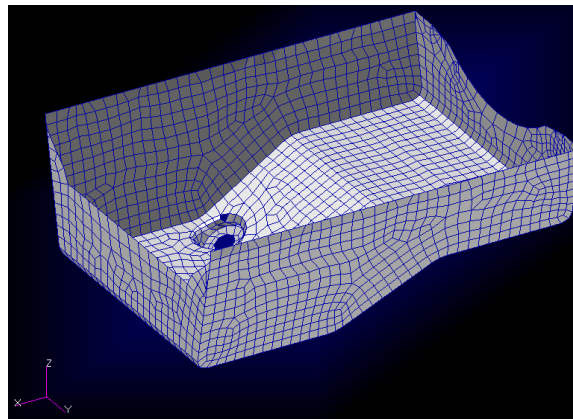
- Topometry

- Modify each thickness
- Special form of sizing optimization



- Topography

- Modify flat to dimpled
- Special form of shape optimization



Basic Optimization Problem Statement

- **Design variables** Find $\{X\} = (X_1, X_2, \dots, X_n)$
- **Objective function:**
Minimize $F(X)$
- **Subject to:**
 - Inequality constraints: $G_j(X) \leq 0 \quad j = 1, 2, \dots, M$
 - Equality constraints: $H_k(X) = 0 \quad k = 1, 2, \dots, L$
 - Side constraints: $X_i^L \leq X_i \leq X_i^U \quad j = 1, 2, \dots, N$

Design Optimization in MSC Nastran (cont.)

- In the example below you optimize two static load cases in subcases 1 and 2, a modal response in subcase 3, and a transient response in subcase 4.

```
SOL 200
CEND
SPC = 100
DESOBJ(MIN) = 15
ANALYSIS = STATICS
SUBCASE 1
  SUBTITLE = static load 1
  DESSUB = 10
  displ = all
  stress = all
  LOAD = 1
SUBCASE 2
  SUBTITLE = static load 2
  DESSUB = 20
  displ = all
  strain(fiber) = all
  LOAD = 2
SUBCASE 3
  SUBTITLE = modal analysis
  ANALYSIS = MODES
  DESSUB = 30
  METHOD = 3
SUBCASE 4
  SUBTITLE = transient analysis
  ANALYSIS = MTRAN
  DESSUB = 40
  METHOD = 4
  DLOAD = 4
  TSTEP = 5
BEGIN BULK
```

Commonly Used Optimization Bulk Data Entries

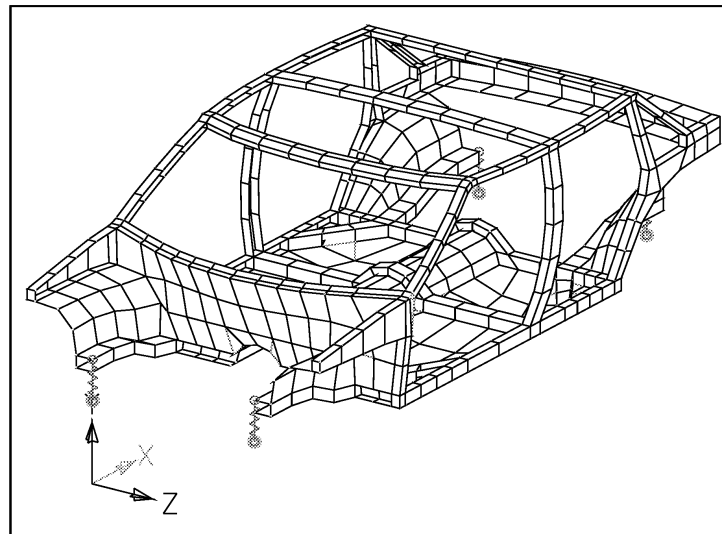
- **DESVAR** Defines a design variable.
- **DVPREL1** Defines the relation between an analysis model property and design variables.
- **DLINK** Relates one design variable to one or more other design variables.
- **DRESP1** Defines a set of direct structural responses that is used in the design either as constraints (referenced by the DCONSTR Bulk Data entry) or as an objective (referenced by the DESOBJ Case Control Command).
- **DCONSTR** Defines a design constraint (referenced by the DESSUB Case Control Command).
- **DCONADD** Defines the design constraints for a subcase as a union of DCONSTR entries.
- **DRESP2** Defines a synthesized response that are used in the design. This response can be either a constraint or an objective.
- **DEQATN** Defines equation(s) for use in design sensitivity.
- **DVCREL1** Defines the relation between a connectivity property and design variables.
- **DVCREL2** Defines the relation between a connectivity property and design variables using a user-supplied equation.
- **DVMREL1** Defines the relation between a material property and design variables.
- **DVMREL2** Defines the relation between a material property and design variables with a user-supplied equation.

Direct Specification of Commonly Used Functions

- **Direct Support of Commonly-used Functions on the DRESP1 entry (e.g, SUM, RSS, AVG, etc.) for transient and frequency response analysis.**
- **Example:**
 - DRESP1,100,RSSCAL,FRDISP,,,3,RSS,100
 - The above DRESP1 performs a root sum square of the displacement at grid point 100, component 3, across the whole forcing frequency range.

Case Study - Optimization Using Frequency Response

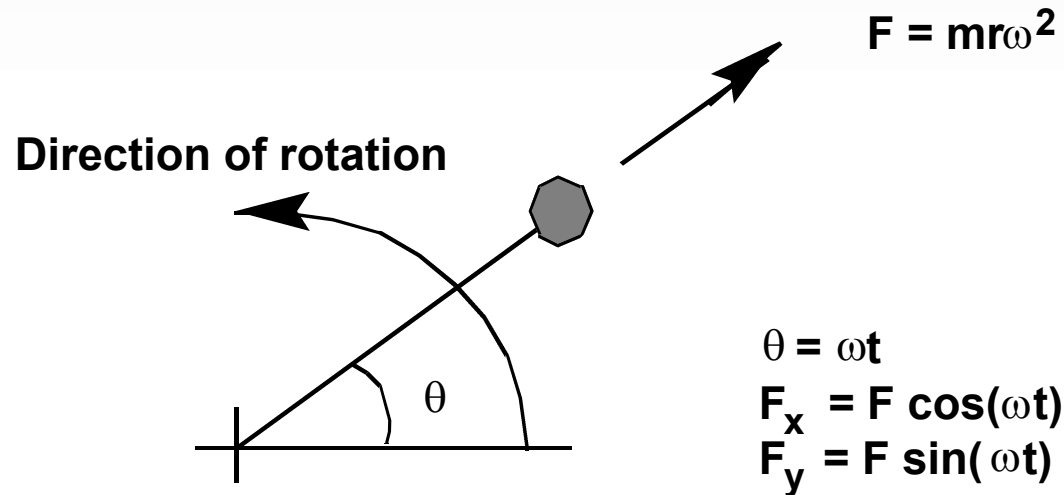
- The automobile model shown below has the front left wheel out of balance. The amount of mass is .3 units and the radius from the center of the wheel to the mass is 10 inches. The frequency range of interest is from 0.5-50 Hz. The tire must not displace by more than 0.5 inches at any frequency. The driver's seat must not displace by more than 0.25 inches between 0.5 and 25 hz. The goal is to minimize the SRSS response of the driver's seat across the whole forcing frequency range. Use the modal method.



Case Study - Optimization Statement

- **Design Variables:**
 - The springs and dampers of the car
- **Objective function:**
 - Minimize the SRSS response at the driver's seat
- **Subject to the following constraints:**
 - The maximum vertical displacement of the tire must be less than 0.5 inches
 - The maximum vertical displacement of the driver seat must be less than 0.25 inches

Case Study - Dynamic Load Input



- Use DAREA to describe magnitude of F_x and F_y (mr)
- Use one RLOAD1 entry for each loading
- Use a DPHASE to describe the phase relationship
- Use a TABLED4 to describe the frequency relationship of the loading (ω^2)
- Use a DLOAD entry to combine the loadings (RLOAD1s)

Case Study - TABLED4 Entry

$$Y = \sum_{i=0}^N A_i \left(\frac{(X - X_1)}{X_2} \right)^i$$

- Set $X_1 = 0$. $X_2 = 1$. $X_3 = 0$. $X_4 = 1000$. (More than highest freq of interest)
- Since X is in cycles per unit time, it must be multiplied by 2π to get radians per unit time - enter 2π
- The input force is $mr\omega^2$ - we will enter mr using DAREA entries, but need the ω^2 term
- We want to use the second (or the ω^2) term from the TABLED4, This can be accomplished by setting $A_0=0.0$ and $A_1=(2\pi)^2$

Dynamic Response Sensitivity

- **Allows calculation of sensitivities of selected dynamic responses with respect to changes in design variables dr_i/dx_j .**
- **These quantities provide insight on which areas are most effective in driving the design.**
- **Sensitivity coefficients can be requested with the DSAPRT case control command**
- **Analysis disciplines:**
 - Direct frequency response
 - Modal frequency response
 - Modal transient response

Description of the Design Model

- **Identify structural properties to be related to design variables (DVPREL1, DVPREL2).**
- **Identify response quantities of interest (DRESP1, DRESP2).**
- **Specify bounds on responses and (optionally) screening criteria (DCONSTR, DSCREEN).**
- **Select output frequencies or time steps in Case Control (OFREQ or OTIME sets).**

Case Study - Input file for Optimization Model

```
$
$ input file to optimize (minimize) the response of a car to a
$ rotating imbalance - V68 - June, 1994
$ use modal approach - up to 100 hz
$
SOL 200
diag 8
CEND
TITLE = Sample dynamic analysis model
SUBTITLE = Rotating force due to tire out of balance
LABEL = perform optimization to minimize driver response
set 999 = 358,471
DISP(phase) = 999
SUBCASE 1
  ANALYSIS = MFREQ
  DESSUB = 100 $ constraints
  DESOBJ(min) = 300 $ design objective - minimize driver response
  DLOAD = 1
  METHOD = 1
  FREQ = 1
BEGIN BULK
eigr1,1,0.,100.
doptprm,desmax,25
include 'car.blk'
include 'springs.blk'
include 'optim1.blk'
param,post,0
$
```

```
$
$ DATA RELATED TO FREQUENCY RESPONSE
$
DLOAD 1 1. 1. 11 1. 12
RLOAD1 11 20 111
RLOAD1 12 30 40 111
DPHASE 40 358 2 90.
DAREA 20 358 1 3.
DAREA 30 358 2 3.
TABLED4 111 0. 1. 0. 100.
0. 39.478 ENDT
FREQ1 1 .5 .5 100
$
ENDDATA
```

Case Study - Partial Listing of car.blk

```
$
$ file - car.blk
$
$ MODEL COURTESY LAPCAD ENGINEERING
$ CHULA VISTA, CALIFORNIA
$
GRID      1      79.0000 56.0000-2.00000
GRID      2      157.000 40.1000-1.30000
.
.
CQUAD4    29      2      22      19      12      21 0.00E+0 0.00E+0
CQUAD4    30      2      21      20      11      22 0.00E+0 0.00E+0
.
.
PSHELL    1      1      0.025      1
PSHELL    2      1      0.200      1
.
.
MAT1      11.000E+73.759E+63.300E-12.600E-41.370E-57.000E+1.3
$
$ end of car.blk
$
```

Case Study - Listing of springs.blk

```
$
$ springs.blk
$
CONM2 3001 1402 1.E8
CONM2 3002 1825 1.E8
CONM2 3003 1358 1.E8
CONM2 3004 1869 1.E8
$
GRID 1402 159.870 11.0000 -14.3750 13456
GRID 1358 20.6116 11.0000 -15.1000 13456
GRID 1825 20.6116 11.0000 -54.9000 13456
GRID 1869 159.870 11.0000 -55.6250 13456
$
CELAS2 1001 10000. 402 1 1402 1
CROD 1002 1002 402 1402
CELAS2 1003 10000. 402 3 1402 3
CELAS2 1011 10000. 825 1 1825 1
CROD 1012 1001 825 1825
CELAS2 1013 10000. 825 3 1825 3
CELAS2 1021 10000. 358 1 1358 1
CROD 1022 1001 358 1358
CELAS2 1023 10000. 358 3 1358 3
CELAS2 1031 10000. 869 1 1869 1
CROD 1032 1002 869 1869
CELAS2 1033 10000. 869 3 1869 3
$
$ properties for springs
$
PROD 1001 1000 1000.
PROD 1002 1000 800.
$ select material so that value of PROD is spring stiffness,
$ therefore, E = I = 10.
MAT1 1000 10.
```

```
$
$ add dampers for shock absorbers
$ front
cvisc 2011 2001 825 1825
cvisc 2021 2001 358 1358
$ back
cvisc 2001 2002 402 1402
cvisc 2031 2002 869 1869
$ damper properties
pvisc 2001 10. 0.
pvisc 2002 5. 0.
$
$ end of springs.blk
$
```

Case Study - Listing of optim1.blk

Define Design Variables

```
desvar,1,frntdamp,1.,.1,10.  
desvar,2,reardamp,1.,.2,20.  
desvar,3,frntstif,1.,.4,2.  
desvar,4,rearstif,1.,.5,2.5
```

Link Design Variables to Properties

```
$  
$ relation between properties and variables  
$  
dvprel1,101,pvisc,2001,3,1.,,,,+dv101  
+dv101,1,10.  
dvprel1,102,pvisc,2002,3,1.,,,,+dv102  
+dv102,2,5.  
dvprel1,103,prod,1001,4,4.,,,,+dv103  
+dv103,3,10.  
dvprel1,104,prod,1002,4,4.,,,,+dv104  
+dv104,4,8.
```

Define Design Constraints

```
$ require that maximum tire displacement be .5 inches  
$  
dconstr,101,200,-.5,.5  
$  
$ require that maximum driver displacement be .25 inches  
$  
dconstr,102,201,-.25,.25,0.5,25.  
$ combine constraints into set 100  
$  
dconadd,100,101,102
```


Case Study - Listing of optim1.blk (cont.)

Select Response Quantities

```
$ select displacement Y at driver seat and mount point as  
$ response quantities  
$  
$ mount point  
$  
dresp1,200,disp,frdisp,,,2,,358  
$  
$ define driver's seat disp as a response  
$  
dresp1,201,driver,frdisp,,,2,,471
```

Define Objective Function - SRSS of Driver Response

```
dresp1,300,srss,frdisp,,,2,RSS,471
```

Case Study - Output

S U M M A R Y O F D E S I G N C Y C L E H I S T O R Y

(HARD CONVERGENCE ACHIEVED)

NUMBER OF FINITE ELEMENT ANALYSES COMPLETED 11
NUMBER OF OPTIMIZATIONS W.R.T. APPROXIMATE MODELS 10

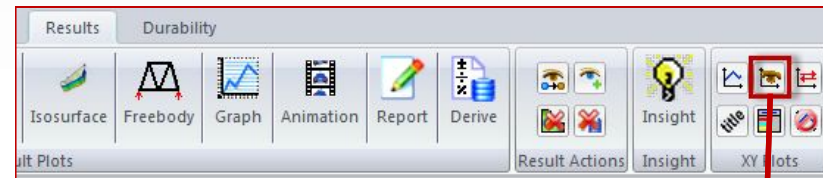
OBJECTIVE AND MAXIMUM CONSTRAINT HISTORY

CYCLE NUMBER	OBJECTIVE FROM APPROXIMATE OPTIMIZATION	OBJECTIVE FROM EXACT ANALYSIS	FRACTIONAL ERROR OF APPROXIMATION	MAXIMUM VALUE OF CONSTRAINT
INITIAL		3.596112E+00		2.824449E+00
1	2.923577E+00	2.825346E+00	3.476772E-02	1.566393E+00
2	2.250654E+00	2.180674E+00	3.209095E-02	7.119287E-01
3	1.716611E+00	1.678514E+00	2.269716E-02	2.069045E-01
4	1.509228E+00	1.505072E+00	2.761246E-03	9.193659E-03
5	1.363270E+00	1.361666E+00	1.177765E-03	-1.417413E-01
6	1.253422E+00	1.250843E+00	2.061693E-03	-2.861117E-01
7	1.173337E+00	1.172155E+00	1.008365E-03	-4.040392E-01
8	1.122992E+00	1.123213E+00	-1.964511E-04	-4.759710E-01
9	1.098395E+00	1.099975E+00	-1.436072E-03	N/A
10	1.099507E+00	1.100117E+00	-5.542642E-04	N/A

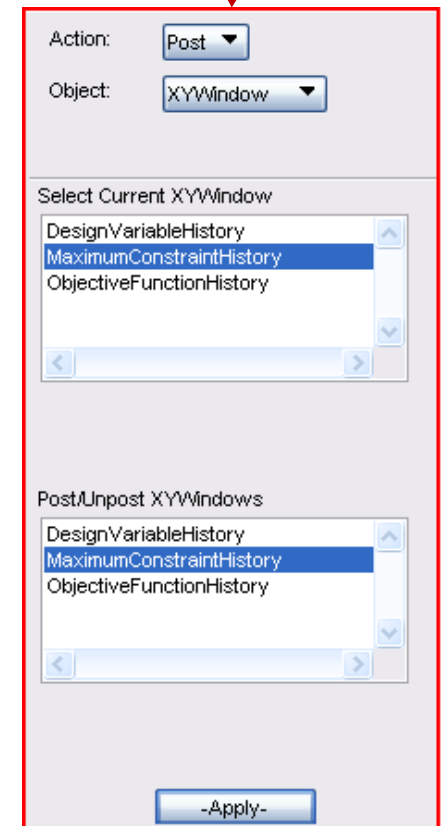
Case Study – Output (cont.)

DESIGN VARIABLE HISTORY														
INTERNAL DV. ID.	EXTERNAL DV. ID.	LABEL	INITIAL	:	1	:	2	:	3	:	4	:	5	:
1	1	FRNTDAMP	1.0000E+00	:	1.5000E+00	:	2.2499E+00	:	3.3538E+00	:	4.0096E+00	:	4.8115E+00	:
2	2	REARDAMP	1.0000E+00	:	1.5000E+00	:	2.2209E+00	:	2.4292E+00	:	2.9053E+00	:	2.9911E+00	:
3	3	FRNTSTIF	1.0000E+00	:	9.9963E-01	:	1.4994E+00	:	1.5017E+00	:	1.5316E+00	:	1.5347E+00	:
4	4	REARSTIF	1.0000E+00	:	9.2901E-01	:	5.0000E-01	:	5.0000E-01	:	5.0000E-01	:	5.0000E-01	:
INTERNAL DV. ID.	EXTERNAL DV. ID.	LABEL	6	:	7	:	8	:	9	:	10	:	11	:
1	1	FRNTDAMP	5.7740E+00	:	6.9288E+00	:	8.3146E+00	:	1.0000E+01	:	1.0000E+01	:		:
2	2	REARDAMP	3.5958E+00	:	4.3150E+00	:	5.1780E+00	:	6.2136E+00	:	7.4563E+00	:		:
3	3	FRNTSTIF	1.5311E+00	:	1.5298E+00	:	1.5290E+00	:	1.5273E+00	:	1.5220E+00	:		:
4	4	REARSTIF	5.0000E-01	:	5.0000E-01	:	5.0000E-01	:	5.0000E-01	:	5.0000E-01	:		:

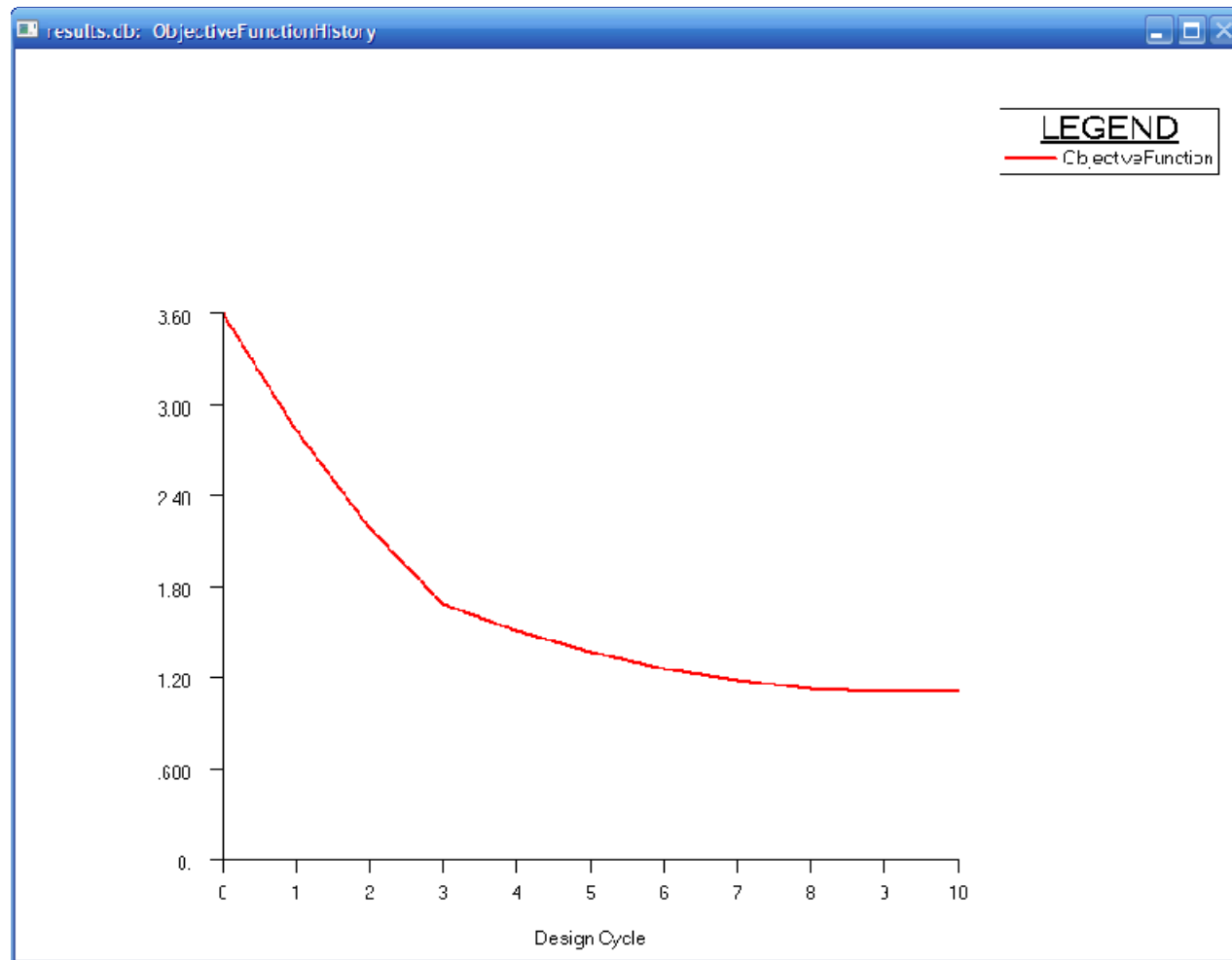
Case Study - Results of Optimization



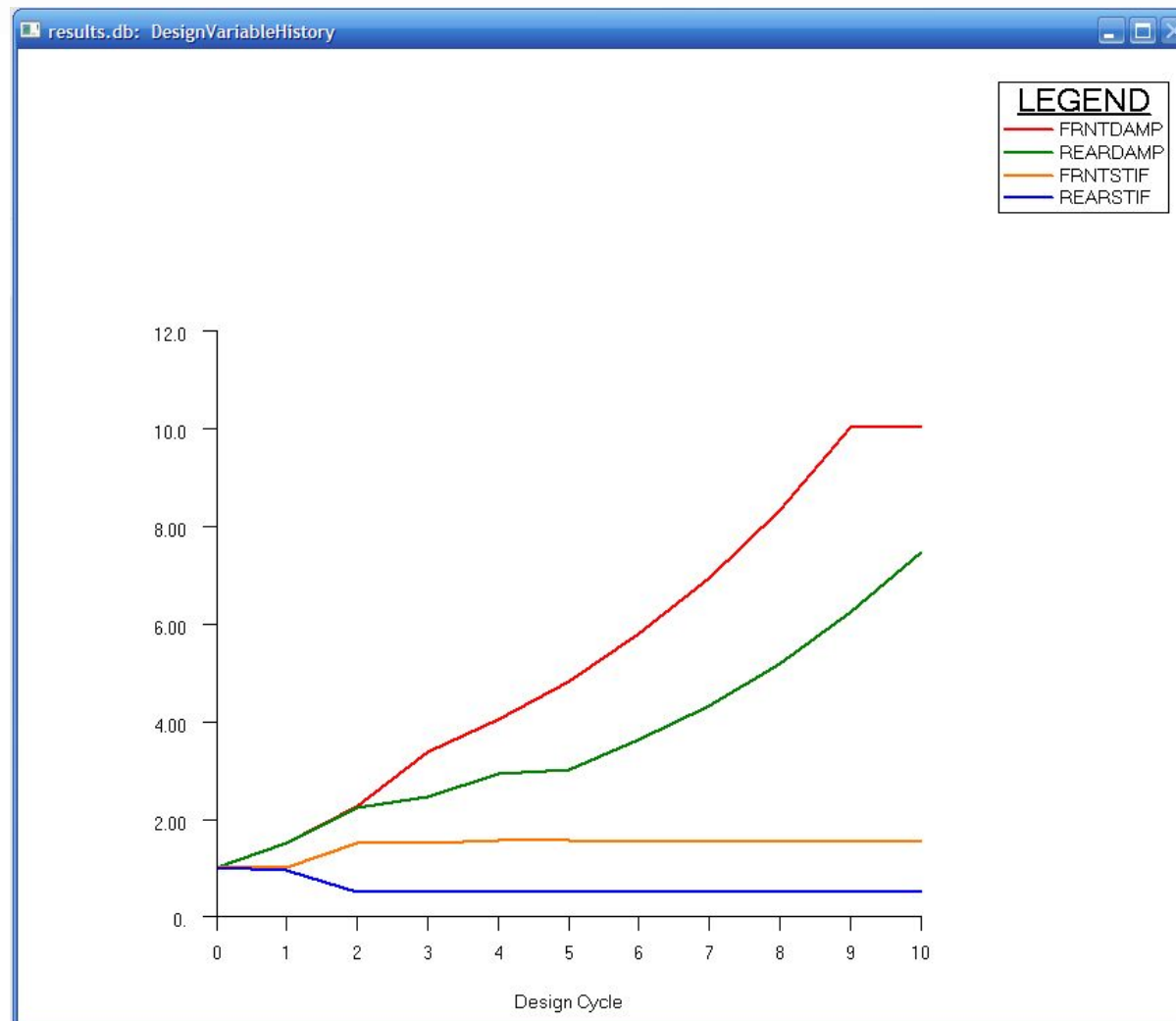
- **Design criteria was satisfied in 10 cycles.**
- **Results follow on the next few pages**
 - Note that MSC Patran was used to create the plots. Even though this isn't a Patran-based class, students have often requested how to obtain these Objective Function and Design History plots. The process is described here.
 - .OP2 file is required because it contains the additional info for XY graphs (auto generated)... add PARAM,POST,-1 to the input file.
 - In Patran, after importing the op2, go to Results, then to the XYPLOT subform and set the Action to Post and Object to XYWindow... the plots are already there and just need selecting.
 - The Frequency Response results are just standard post-processing in Patran's Results From (XYGRAPH)



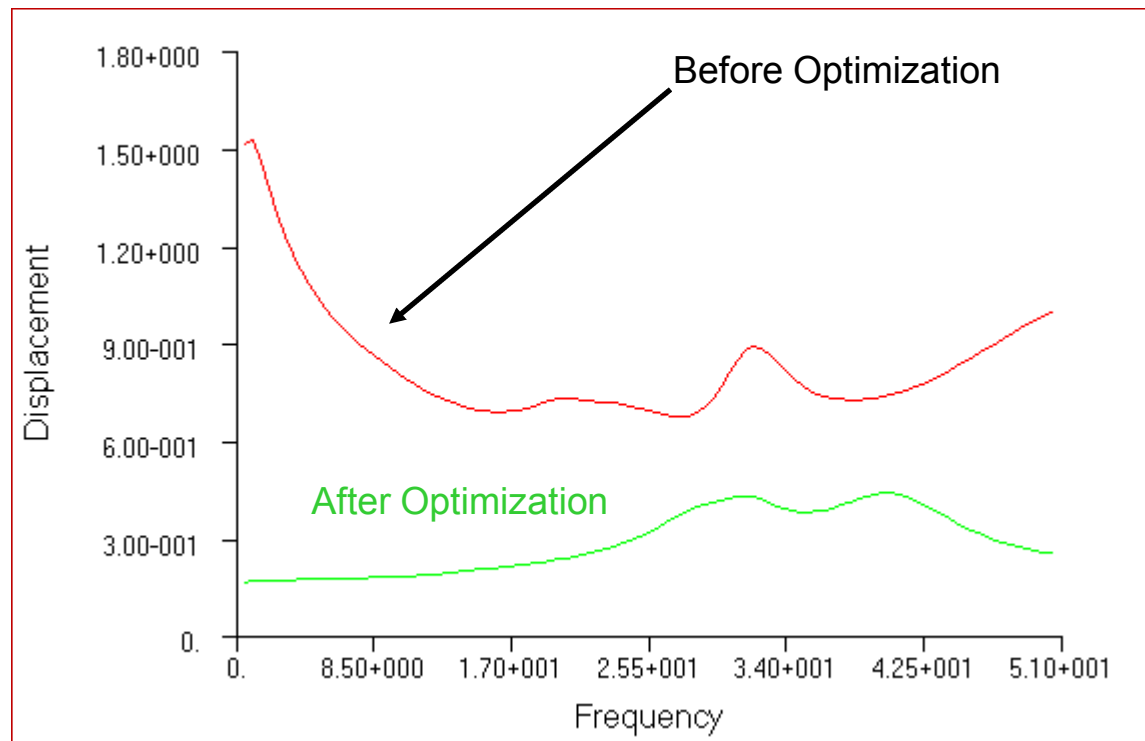
Case Study - Objective Function History



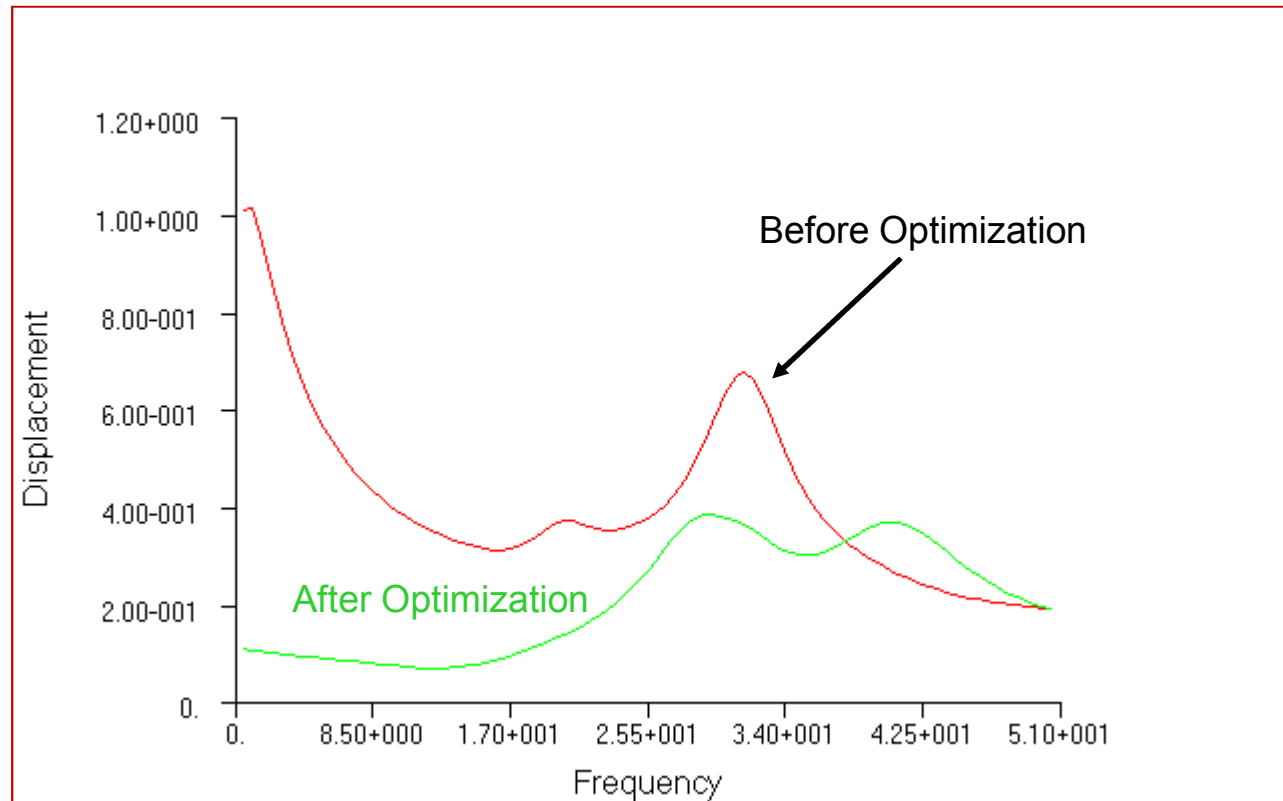
Case Study - Design Variable History



Case Study - Wheel Response



Driver's Seat Response



Conclusion

- **With minimum effort, the design is modified to satisfy constraints and minimize selected responses.**
- **SOL 200 is a valuable design tool for dynamic analysis.**

WORKSHOP 8

OPTIMIZATION USING NORMAL MODES

SECTION 16

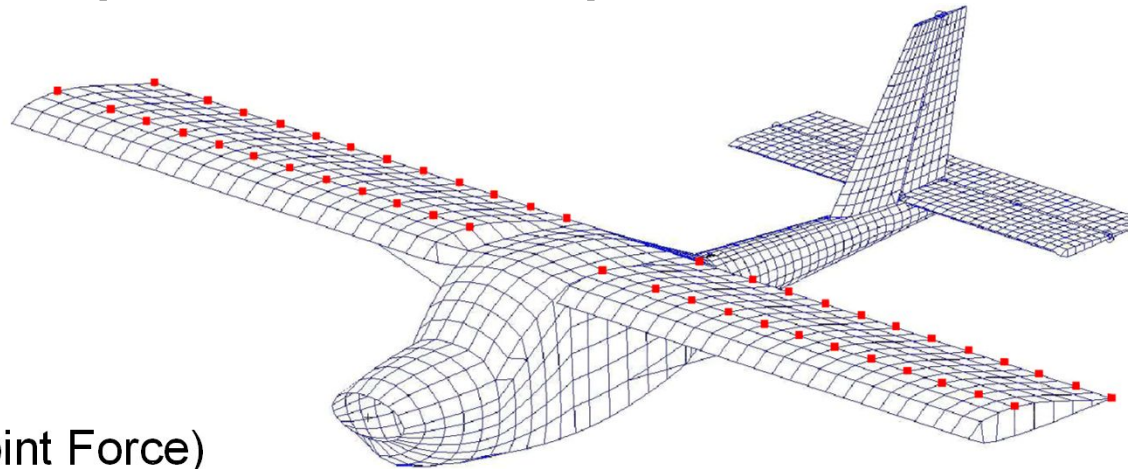
Monitor Points

Covered in this Section

- **What are Monitor Points**
- **Types of Monitor Point**
- **MONPNT1**
- **MONPNT2**
- **MONPNT3**
- **MONDSP1**
- **Combining Monitor Point Outputs – MONSUM**
- **MONITOR – Case Control Command**

What are Monitor Points

- **Monitor points allow output of certain key quantities, & combinations of like output quantities, to be output in static and dynamic analysis**
 - Displacement
 - Load
 - Stress
 - Force
 - Strain
 - Free body forces (Grid Point Force)
- **The outputs are in addition to the usual case control output**
- **Explicit user defined labels (up to 64 characters)**
- **Typically these outputs will be key quantities for the analysis**



Types of Monitor Point

- **There are 4 basic types of monitor point**
- **MONPNT1**
 - Summation of applied & inertia based loads about a point
 - Provides the user with a way to extract the applied loading for a specified set of structural nodes (or aerodynamic elements for static or dynamic aeroelasticity).
 - Enables the batch calculation of VMT (shear, moment, and torque) data for user specified regions and locations
- **MONPNT2**
 - Single entity element stress/strain/force output
 - Provides a way of pinpointing a particular response for output, as opposed to finding it in a large OFP

Types of Monitor Point (cont.)

- **There are 4 basic types of monitor point (cont.)**
- **MONPNT3**
 - Summation of free body interface loads (similar to GPFORCE)
 - Provides a summation of the internal loads and therefore, is useful in calculating resultant forces at a cut in the structure
- **MONDSP1**
 - Averaged displacement from a collection of GRID points (similar to RBE3)
 - Ability to provide an averaged displacement is seen as providing a qualitative assessment of the deflection of a structure

Introduction to Monitor Points

- Prior to MSC Nastran 2010, monitor points were only available in SOLs 101, 103, 144 and 146.
- These sections discuss their application in the linear dynamic response solution sequences; i.e., SOLs 108, 109, 111, and 112.
- The *MSC Nastran Quick Reference Guide* provides guidance on specifying the *MONPNT1*, *MONPNT2*, *MONPNT3*, and *MONDSP1* Bulk Data entries.
- The **MONITOR** Case Control command must be used to obtain output results in the dynamic solution sequences. The command provides options for frequency response results in terms of either the Real/Imag or MAG/Phase form.
- The output is limited to the .f06 file in all solution sequences except SOL 146 which also has op2 support.

Guidelines and Limitations

- **Dynamic monitor points are not available as design responses in SOL 200.**
- **Inertia results are available for the MONPNT1 but have not been implemented for the MONPNT3.**
- **The MONSUM feature can be used but is of limited utility when the MONSUM spans monitor types as described in Connectors.**
- **In Frequency Response analysis, the monitor point output is in SORT2 format.**

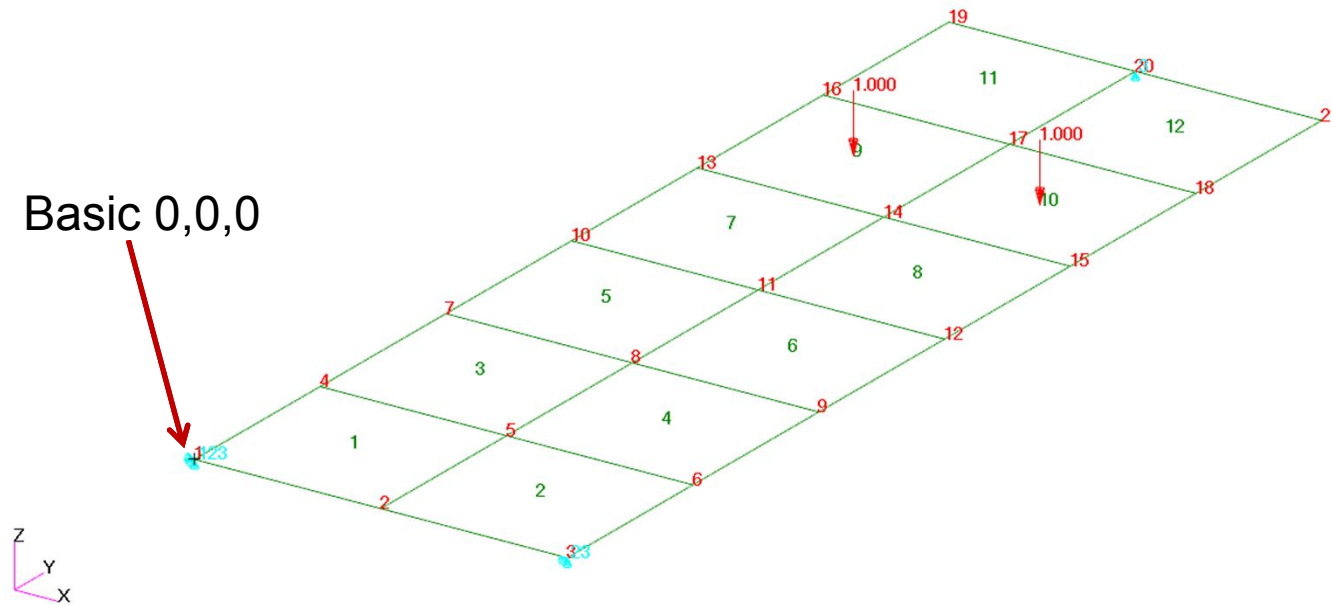
MONPNT1

MONPNT1 Considerations

- **The MONPNT1 is available for SOLs 101, 108, 109, 111, 112, 144, and 146.**
- **The entry can create either an aerodynamic or a structural MONPNT1, depending on the data provided in the COMP field.**
 - A structural MONPNT1 can have the same name as an aerodynamic MONPNT1, but two structural or two aerodynamic MONPNT1s cannot have the same name.
- **In SOL 144, static aeroelasticity, the structural MONDPNT1 integration is modified to account for the redistribution of loads an inertia caused by MPCs and rigid elements.**
 - In all other supported solution sequences, the integration is done using the g-set loads on all grid points specified on the referenced SET1 entry.

MONPNT1 - Statics

- **When used in statics MONPNT1 will sum applied loads**
 - Yields the same output as OLOAD RESULTANT if ALL GRID points are selected and summation is about the same point defined by PARAM,GRDPNT (basic system by default)



MONPNT1 - Statics

- OLOAD RESULTANT & MONPNT1 output compared

RESULTANTS ABOUT ORIGIN OF SUPERELEMENT BASIC COORDINATE SYSTEM IN SUPERELEMENT BASIC SYSTEM COORDINATES.

		OLOAD			RESULTANT		
SUBCASE/ DAREA ID	LOAD TYPE	T1	T2	T3	R1	R2	R3
1	FX	0.000000E+00	----	----	----	0.000000E+00	0.000000E+00
	FY	----	4.000000E+03	----	0.000000E+00	----	2.000000E+03
	FZ	----	----	0.000000E+00	0.000000E+00	0.000000E+00	----
	MX	----	----	----	0.000000E+00	----	----
	MY	----	----	----	----	0.000000E+00	----
	MZ	----	----	----	----	----	0.000000E+00
	TOTALS	0.000000E+00	4.000000E+03	0.000000E+00	0.000000E+00	0.000000E+00	2.000000E+03

STRUCTURAL MONITOR POINT INTEGRATED LOADS (MONPNT1)

MONITOR POINT NAME = STN0 (G) COMPONENT = 30 CLASS = GENERAL SUBCASE NO. 1

LABEL = OLOAD EQUIVALENT ABOUT BASIC

CP = 0 X = 0.00000E+00 Y = 0.00000E+00 Z = 0.00000E+00 CD = 0

AXIS RIGID APPLIED

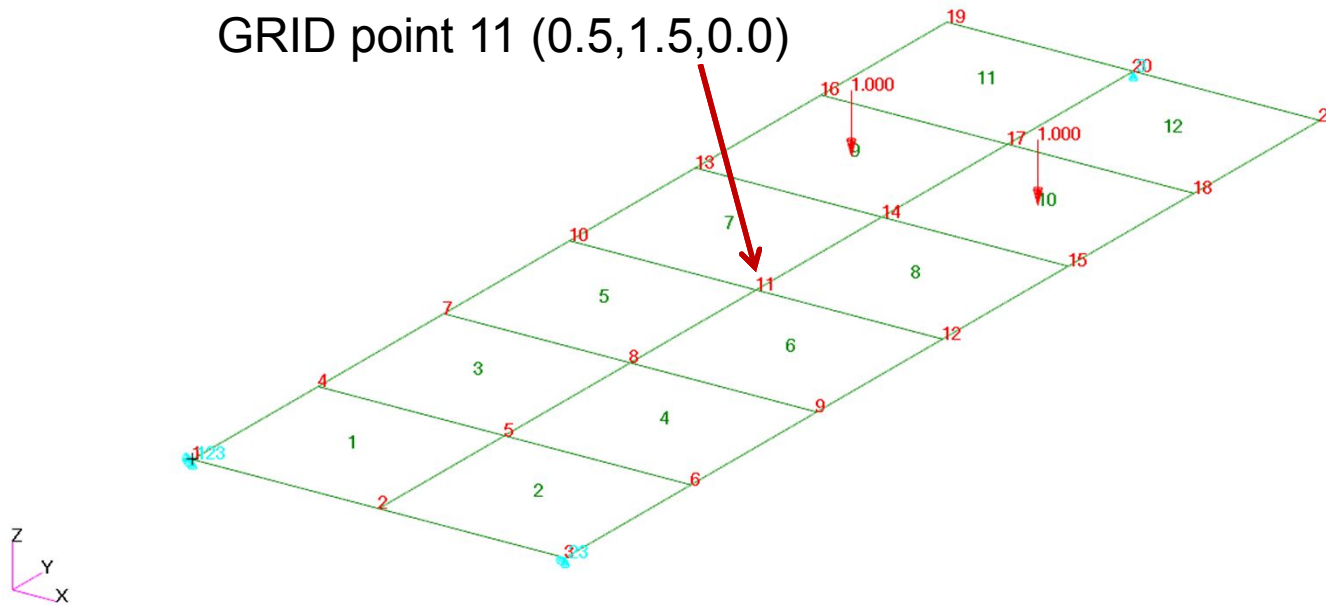
----	-----
CX	0.000000E+00
CY	4.000000E+03
CZ	0.000000E+00
CMX	0.000000E+00
CMY	0.000000E+00
CMZ	2.000000E+03

- The sum of applied loads is reported in the column RIGID APPLIED

MONPNT1 - Statics

- **Change the location of the point**
 - Use PARAM,GRDPNT to define a different point for OLOAD RESULTANT output
 - For example, PARAM,GRDPNT,11

GRID point 11 (0.5,1.5,0.0)



MONPNT1 - Statics

- OLOAD RESULTANT & MONPNT1 output compared**

RESULTANTS ABOUT 11 IN SUPERELEMENT BASIC SYSTEM COORDINATES.							
		OLOAD			RESULTANT		
SUBCASE/ DAREA ID	LOAD TYPE	T1	T2	T3	R1	R2	R3
1	FX	0.000000E+00	----	----	----	0.000000E+00	0.000000E+00
	FY	----	4.000000E+03	----	0.000000E+00	----	0.000000E+00
	FZ	----	----	0.000000E+00	0.000000E+00	0.000000E+00	----
	MX	----	----	----	0.000000E+00	----	----
	MY	----	----	----	----	0.000000E+00	----
	MZ	----	----	----	----	----	0.000000E+00
	TOTALS	0.000000E+00	4.000000E+03	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00

S T R U C T U R A L M O N I T O R P O I N T I N T E G R A T E D L O A D S (MONPNT1)

MONITOR POINT NAME = STN4(G) COMPONENT = 31 CLASS = GENERAL SUBCASE NO. 1

LABEL = OLOAD EQUIVALENT ABOUT MODEL CENTRE

CP = 0 X = 5.00000E-01 Y = 1.50000E+00 Z = 0.00000E+00 CD = 0

AXIS	RIGID APPLIED
----	-----
CX	0.000000E+00
CY	4.000000E+03
CZ	0.000000E+00
CMX	0.000000E+00
CMY	0.000000E+00
CMZ	0.000000E+00

MONPNT1 - Statics

- The output is obtained by using the MONPNT1 bulk data entry

1	2	3	4	5	6	7	8	9	10
MONPNT1	NAME	LABEL							
	AXES	COMP	CP	X	Y	Z	CD		

- For example, the output above was obtained using the following MONPNT1 entries

```

MONPNT1  STN0 (G)  OLOAD EQUIVALENT ABOUT BASIC
          123456  30
$
MONPNT1  STN4 (G)  OLOAD EQUIVALENT ABOUT MODEL CENTRE
          123456  30      0.0      0.5      1.5
$
SET1      30      1      THRU      21
    
```

MONPNT1 - Statics

- AXES determines which components are reported

1	2	3	4	5	6	7	8	9	10
MONPNT1	NAME	LABEL							
	AXES	COMP	CP	X	Y	Z	CD		

MONPNT1 STN0 (G) OLOAD EQUIVALENT ABOUT BASIC
 123456 30

\$

SET1 30 1 THRU 21

STRUCTURAL MONITOR POINT INTEGRATED LOADS (MONPNT1)

MONITOR POINT NAME = STN0 (G) COMPONENT = 30 CLASS = GENERAL SUBCASE NO. 1
 LABEL = OLOAD EQUIVALENT ABOUT BASIC
 CP = 0 X = 0.000000E+00 Y = 0.000000E+00 Z = 0.000000E+00 CD = 0

AXIS	RIGID APPLIED
CX	0.000000E+00
CY	4.000000E+03
CZ	0.000000E+00
CMX	0.000000E+00
CMY	0.000000E+00
CMZ	2.000000E+03

MONPNT1 - Statics

- COMP selects the SET1 to define the GRID points for summation

1	2	3	4	5	6	7	8	9	10
MONPNT1	NAME	LABEL							
	AXES	COMP	CP	X	Y	Z	CD		

MONPNT1 STN0 (G) OLOAD EQUIVALENT ABOUT BASIC
 123456 30

\$
 SET1 30 1 THRU 21

STRUCTURAL MONITOR POINT INTEGRATED LOADS (MONPNT1)

MONITOR POINT NAME = STN0 (G) COMPONENT = 30 CLASS = GENERAL SUBCASE NO. 1
 LABEL = OLOAD EQUIVALENT ABOUT BASIC
 CP = 0 X = 0.000000E+00 Y = 0.000000E+00 Z = 0.000000E+00 CD = 0

AXIS	RIGID APPLIED
CX	0.000000E+00
CY	4.000000E+03
CZ	0.000000E+00
CMX	0.000000E+00
CMY	0.000000E+00
CMZ	2.000000E+03

MONPNT1 - Statics

- CP is the coordinate system in which X,Y,Z are defined

1	2	3	4	5	6	7	8	9	10
MONPNT1	NAME	LABEL							
	AXES	COMP	CP	X	Y	Z	CD		

```

MONPNT1  STN0 (G)  OLOAD EQUIVALENT ABOUT BASIC
          123456  30
$
SET1      30      1      THRU      21
    
```

```

S T R U C T U R A L   M O N I T O R   P O I N T   I N T E G R A T E D   L O A D S (MONPNT1)

MONITOR POINT NAME = STN0 (G)      COMPONENT = 30      CLASS = GENERAL      SUBCASE NO.  1
LABEL = OLOAD EQUIVALENT ABOUT BASIC
CP = 0      X = 0.000000E+00      Y = 0.000000E+00      Z = 0.000000E+00      CD = 0

  AXIS      RIGID APPLIED
  ----      -
  CX      0.000000E+00
  CY      4.000000E+03
  CZ      0.000000E+00
  CMX      0.000000E+00
  CMY      0.000000E+00
  CMZ      2.000000E+03
    
```

- Blank for CP = 0
(the basic coordinate system)

MONPNT1 - Statics

- X,Y,Z coordinates (in CP) of the summation location

1	2	3	4	5	6	7	8	9	10
MONPNT1	NAME	LABEL							
	AXES	COMP	CP	X	Y	Z	CD		

MONPNT1 STN0 (G) OLOAD EQUIVALENT ABOUT BASIC
123456 30

\$

SET1 30 1 THRU 21

S T R U C T U R A L M O N I T O R P O I N T I N T E G R A T E D L O A D S (MONPNT1)

MONITOR POINT NAME = STN0 (G) COMPONENT = 30 CLASS = GENERAL SUBCASE NO. 1

LABEL = OLOAD EQUIVALENT ABOUT BASIC

CP = 0 X = 0.000000E+00 Y = 0.000000E+00 Z = 0.000000E+00 CD = 0

AXIS RIGID APPLIED

----	-----
CX	0.000000E+00
CY	4.000000E+03
CZ	0.000000E+00
CMX	0.000000E+00
CMY	0.000000E+00
CMZ	2.000000E+03

- Does same job as PARAM,GRDPNT for the OLOAD RESULTANT except output may be in ANY coordinate system

MONPNT1 - Statics

- CD defines the output coordinate system

1	2	3	4	5	6	7	8	9	10
MONPNT1	NAME	LABEL							
	AXES	COMP	CP	X	Y	Z	CD		

MONPNT1 STN0 (G) OLOAD EQUIVALENT ABOUT BASIC
123456 30

\$

SET1 30 1 THRU 21

STRUCTURAL MONITOR POINT INTEGRATED LOADS (MONPNT1)

MONITOR POINT NAME = STN0 (G) COMPONENT = 30 CLASS = GENERAL SUBCASE NO. 1

LABEL = OLOAD EQUIVALENT ABOUT BASIC

CP = 0 X = 0.000000E+00 Y = 0.000000E+00 Z = 0.000000E+00

CD = 0

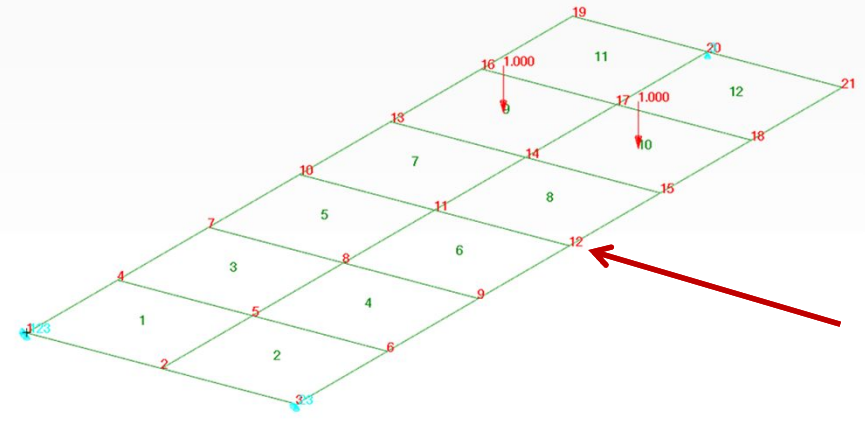
AXIS RIGID APPLIED

```

-----
CX 0.000000E+00
CY 4.000000E+03
CZ 0.000000E+00
CMX 0.000000E+00
CMY 0.000000E+00
CMZ 2.000000E+03
  
```

MONPNT1 - Statics

- Now we can sum applied loads about any point over any domain of GRID points
- For example, calculate applied load summation for **GRID** points 10,11,12 relative to the center of the plate



```

MONPNT1  STN4 (H)  INTEGRATED LOADS AT MID-SPAN GRIDS
          123456  34          0.5      1.5      0.0
SET1      34      10      11      12
    
```

```

S T R U C T U R A L   M O N I T O R   P O I N T   I N T E G R A T E D   L O A D S   ( M O N P N T 1 )

MONITOR POINT NAME = STN4 (H)          COMPONENT = 34          CLASS = GENERAL          SUBCASE NO.    1
LABEL = INTEGRATED LOADS AT MID-SPAN GRIDS
CP =      0      X = 5.00000E-01      Y = 1.50000E+00      Z = 0.00000E+00      CD =      0

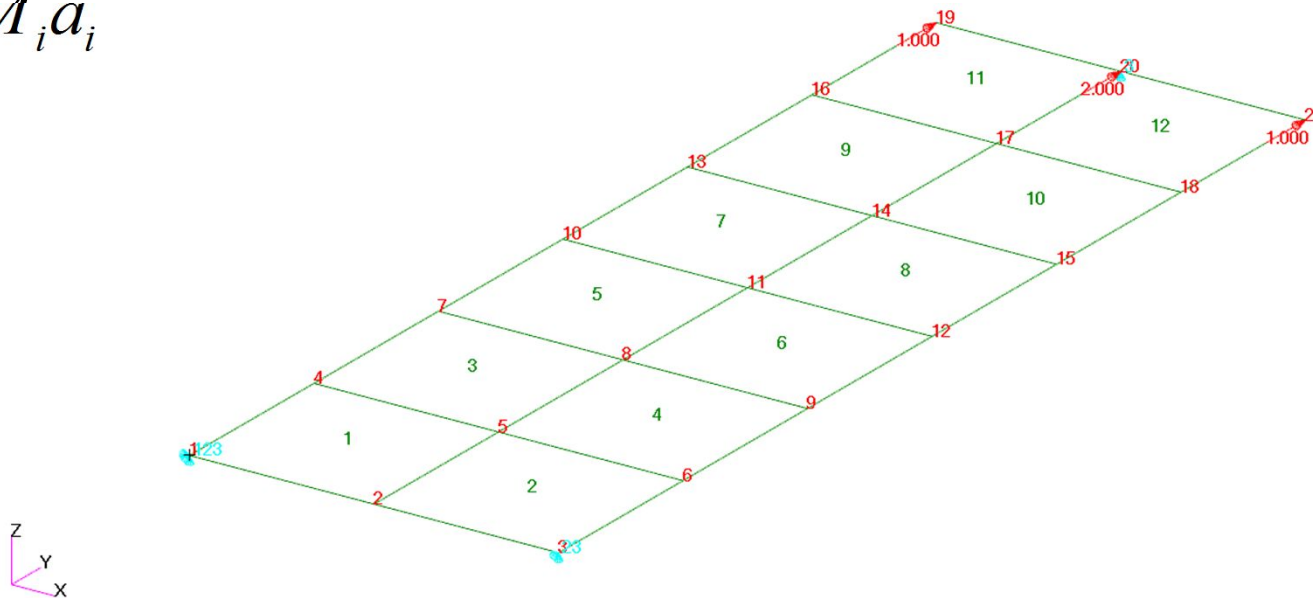
  AXIS      RIGID APPLIED
  ----      -
  CX      0.000000E+00
  CY      0.000000E+00
  CZ      0.000000E+00
  CMX     0.000000E+00
  CMY     0.000000E+00
  CMZ     0.000000E+00
    
```

- The result of zero for all 6 directions just means no loads are applied to any of the GRID points in the selected SET1

MONPNT1 - Dynamics

- When used in dynamics MONPNT1 will report the sum of applied load *and* the sum of inertia loads
- Inertial load is calculated by summing the mass * acceleration terms over the degrees of freedom of the specified GRID points

$$\sum_i^N M_i a_i$$



MONPNT1 - Dynamics

- Using the same MONPNT1 entry we defined for statics

1	2	3	4	5	6	7	8	9	10
MONPNT1	NAME	LABEL							
	AXES	COMP	CP	X	Y	Z	CD		

```

MONPNT1  STN0 (G)  OLOAD EQUIVALENT ABOUT BASIC
          123456  30
$
SET1      30      1      THRU      21
    
```

- The applied loads are calculated as for statics, over all GRID points in all directions
- The inertial loads are calculated by multiplying the mass at each GRID point/direction by the acceleration and summing over the GRID points

MONPNT1 - Dynamics

- You can ignore the FLEXIBLE INCREMENT, GUST and TOTAL AERO columns
 - These are only used in aero-elastic analysis

```

S T R U C T U R A L   M O N I T O R   P O I N T   I N T E G R A T E D   L O A D S (MONPNT1)
                        (REAL/IMAGINARY)

MONITOR POINT NAME = STN0 (G)      COMPONENT = CY      GENERAL      SUBCASE NO.      1
LABEL = OLOAD EQUIVALENT ABOUT BASIC
CP =      0      X = 0.000000E+00      Y = 0.000000E+00      Z = 0.000000E+00      CD =      0

      FREQUENCY      INERTIAL      EXTERNAL      FLEXIBLE      GUST      TOTAL      TOTAL
      -----      -----      -----      INCREMENT      -----      AERO      -----
1.000000E+00      4.515723E-02      4.000000E+03      -----      -----      0.000000E+00      4.000045E+03
                        -2.900689E-06      0.000000E+00      -----      -----      0.000000E+00      -2.900689E-06
    
```

MONPNT1 - Dynamics

- OLOAD RESULTANT & MONPNT1 EXTERNAL output compared**

RESULTANTS ABOUT ORIGIN OF SUPERELEMENT BASIC COORDINATE SYSTEM IN SUPERELEMENT BASIC SYSTEM COORDINATES.

		OLOAD			RESULTANT		
SUBCASE/ DAREA ID	LOAD TYPE	T1	T2	T3	R1	R2	R3
1	FX	0.000000E+00	----	----	----	0.000000E+00	0.000000E+00
	FY	----	4.000000E+03	----	0.000000E+00	----	2.000000E+03
	FZ	----	----	0.000000E+00	0.000000E+00	0.000000E+00	----
	MX	----	----	----	0.000000E+00	----	----
	MY	----	----	----	----	0.000000E+00	----
	MZ	----	----	----	----	----	0.000000E+00
	TOTALS	0.000000E+00	4.000000E+03	0.000000E+00	0.000000E+00	0.000000E+00	2.000000E+03

STRUCTURAL MONITOR POINT INTEGRATED LOADS (MONPNT1)
(REAL/IMAGINARY)

MONITOR POINT NAME = STN0 (G) COMPONENT = CY GENERAL SUBCASE NO. 1

LABEL = OLOAD EQUIVALENT ABOUT BASIC

CP = 0 X = 0.000000E+00 Y = 0.000000E+00 Z = 0.000000E+00 CD = 0

FREQUENCY	INERTIAL	EXTERNAL	FLEXIBLE INCREMENT	GUST	TOTAL AERO	TOTAL
1.000000E+00	4.515723E-02	4.000000E+03			0.000000E+00	4.000045E+03
	-2.900689E-06	0.000000E+00			0.000000E+00	-2.900689E-06

MONPNT1 - Dynamics

- MONPNT1 INERTIAL Load

STRUCTURAL MONITOR POINT INTEGRATED LOADS (MONPNT1)
(REAL/IMAGINARY)

MONITOR POINT NAME = STN0 (G) COMPONENT = **CY** GENERAL SUBCASE NO. 1

LABEL = OLOAD EQUIVALENT ABOUT BASIC

CP = 0 X = 0.000000E+00 Y = 0.000000E+00 Z = 0.000000E+00 CD = 0

FREQUENCY	INERTIAL	EXTERNAL	FLEXIBLE INCREMENT	GUST	TOTAL AERO	TOTAL
-----	-----	-----	-----	-----	-----	-----
1.000000E+00	4.515723E-02 -2.900689E-06	4.000000E+03 0.000000E+00			0.000000E+00 0.000000E+00	4.000045E+03 -2.900689E-06

MONPNT2

MONPNT2 Considerations

- The MONPNT2 is available for SOLs 101, 103, 108, 109, 111, 112 and 144.
- Most element types have some items that can be monitored.
- An assumption is made that the desired component is linear with respect to the displacement vector. If this assumption is not valid, the results will be approximate.
- Fictitious Table/Type/NDDLitems/EID generate a warning message and are ignored.
- NDDL descriptions for Table=FORCE can be found in the *MSC Nastran 2013 DMAP Programmer's Guide* within the OEF datablock description. Table=STRESS and STRAIN are contained in the OES datablock description.

MONPNT2

- **MONPNT2 allows result quantities for single items of individual elements to be reported**
- **Outputs the same results as**
 - FORCE (or ELFORCE)
 - STRESS
 - STRAIN
- **For example, monitor the Sigma-X stress in CQUAD4 element 2**
- **Allows “full” data recovery to go to the post processor and obtain spot check values in the f06 file**

MONPNT2 - Statics

- Compare MONPNT2 output with standard output

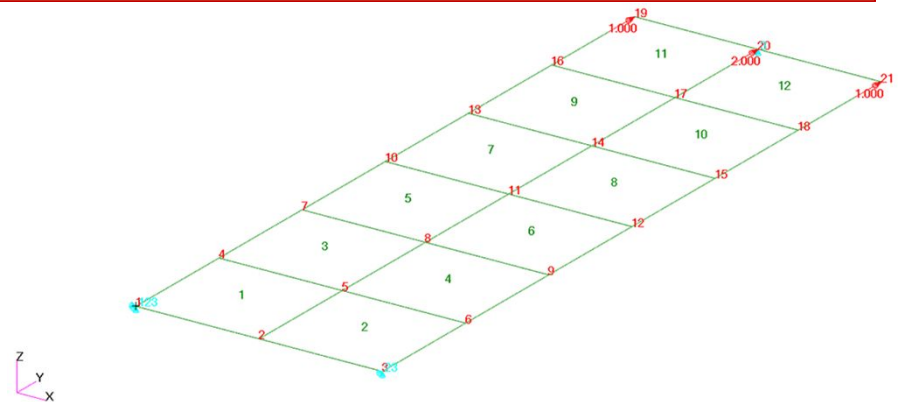
```

STRESSES IN QUADRILATERAL ELEMENTS (QUAD4)
ELEMENT FIBER STRESSES IN ELEMENT COORD SYSTEM PRINCIPAL STRESSES (ZERO SHEAR)
ID. DISTANCE NORMAL-X NORMAL-Y SHEAR-XY ANGLE MAJOR MINOR VON MISES
2 -5.000000E-03 -8.136726E+04 4.000000E+05 -1.730592E+05 -72.1413 4.557588E+05 -1.371260E+05 5.376021E+05
5.000000E-03 -8.136726E+04 4.000000E+05 -1.730592E+05 -72.1413 4.557588E+05 -1.371260E+05 5.376021E+05
    
```

STRUCTURAL INTERNAL MONITOR POINT LOADS (MONPNT2)

MONITOR POINT NAME = MONPNT2

VALUE	EID	ETYPE	CLASS	RESPONSE	LABEL	SUBCASE NO.
-8.136726E+04	2	CQUAD4	STRESS	SX1	SX1 NORMAL IN X AT Z1	1
4.000000E+05	2	CQUAD4	STRESS	SY1	SY1 NORMAL IN Y AT Z1	1
-1.730592E+05	2	CQUAD4	STRESS	TX1	TX1 SHEAR IN XY AT Z1	1



MONPNT2 - Dynamics

- Compare MONPNT2 output with standard output

```

FREQUENCY = 1.000000E+00
COMPLEX STRESSES IN QUADRILATERAL ELEMENTS (QUAD4)
              (REAL/IMAGINARY)

ELEMENT      FIBER
ID.          DISTANCE
  2          -5.000000E-03  -8.136887E+04 / 7.126149E+00 4.000088E+05 / -3.353318E+01 -1.730622E+05 / 1.427360E+01
              5.000000E-03 -8.136887E+04 / 7.126149E+00 4.000088E+05 / -3.353318E+01 -1.730622E+05 / 1.427360E+01
    
```

- STRESSES IN ELEMENT COORDINATE SYSTEM -

NORMAL-X NORMAL-Y SHEAR-XY

STRUCTURAL INTERNAL MONITOR POINT LOADS (MONPNT2)
(REAL/IMAGINARY)

MONITOR POINT NAME = SX1 GENERAL SUBCASE NO. 1
LABEL = NORMAL IN X AT Z1

FREQUENCY	VALUE	EID	ETYPE	CLASS	RESPONSE
1.000000E+00	-8.136887E+04	2	CQUAD4	STRESS	SX1
	7.126149E+00	2	CQUAD4	STRESS	SX1

MONITOR POINT NAME = SY1 GENERAL SUBCASE NO. 1
LABEL = NORMAL IN Y AT Z1

FREQUENCY	VALUE	EID	ETYPE	CLASS	RESPONSE
1.000000E+00	4.000088E+05	2	CQUAD4	STRESS	SY1
	-3.353318E+01	2	CQUAD4	STRESS	SY1

MONPNT2

- The output is obtained by using the MONPNT2 bulk data entry

1	2	3	4	5	6	7	8	9	10
MONPNT2	NAME	LABEL							
	TABLE	TYPE	NDDLitem	EID					

- For example, the output above was obtained using the following MONPNT2 entries

```

MONPNT2  SX1      NORMAL IN X AT Z1
          STRESS  CQUAD4  SX1      2
MONPNT2  SY1      NORMAL IN Y AT Z1
          STRESS  CQUAD4  SY1      2
MONPNT2  TXY1     SHEAR  IN XY AT Z1
          STRESS  CQUAD4  TXY1     2
    
```

MONPNT2

- TABLE defines the type of result quantity

1	2	3	4	5	6	7	8	9	10
MONPNT2	NAME	LABEL							
	TABLE	TYPE	NDDLitem	EID					

```

MONPNT2  SX1  NORMAL IN X AT Z1
          STRESS CQUAD4  SX1    2
MONPNT2  SY1  NORMAL IN Y AT Z1
          STRESS CQUAD4  SY1    2
MONPNT2  TXY1 SHEAR IN XY AT Z1
          STRESS CQUAD4  TXY1   2
    
```

- It is reported in the CLASS column in the output

VALUE	EID	ETYP	CLASS	RESPONSE	LABEL	SUBCASE NO.			
-8.136726E+04	2	CQUAD4	STRESS	SX1	SX1	NORMAL	IN	X	AT Z1 1

MONPNT2

- TYPE is the element type, as per the QRG

1	2	3	4	5	6	7	8	9	10
MONPNT2	NAME	LABEL							
	TABLE	TYPE	NDDLItem	EID					

```

MONPNT2  SX1  NORMAL IN X AT Z1
          STRESS CQUAD4 SX1      2
MONPNT2  SY1  NORMAL IN Y AT Z1
          STRESS CQUAD4 SY1      2
MONPNT2  TXY1 SHEAR IN XY AT Z1
          STRESS CQUAD4 TXY1     2
    
```

- It is reported in the ETYPE column in the output

VALUE	EID	ETYPE	CLASS	RESPONSE	LABEL	SUBCASE NO.				
-8.136726E+04	2	CQUAD4	STRESS	SX1	SX1	NORMAL	IN	X	AT	Z1 1

MONPNT2

- NDDLitem is the individual quantity to output

1	2	3	4	5	6	7	8	9	10
MONPNT2	NAME	LABEL							
	TABLE	TYPE	NDDLitem	EID					

```

MONPNT2  SX1      NORMAL IN X AT Z1
          STRESS  CQUAD4 SX1      2
MONPNT2  SY1      NORMAL IN Y AT Z1
          STRESS  CQUAD4 SY1      2
MONPNT2  TXY1     SHEAR  IN XY AT Z1
          STRESS  CQUAD4 TXY1     2
    
```

- It is reported in the RESPONSE column in the output

VALUE	EID	ETYPE	CLASS	RESPONSE	LABEL						SUBCASE NO.
-8.136726E+04	2	CQUAD4	STRESS	SX1	SX1	NORMAL	IN	X	AT	Z1	1

- The NDDLitem can be found either by looking in the NDDL or consulting the *DMAP Programmer's Guide*
- Look in the section *Data Block Descriptions*
 - Data Block OES for STRESS and STRAIN
 - Datablock OEF for FORCE

OES Table of element stresses or strains

For all analysis types (real and complex) and SORT1 and SORT2 formats.

Record 0 -- HEADER

Word	Name	Type	Description
1	NAME(2)	CHAR4	Data block name
3	WORD	I	No def or month, year, one, one
Word 3 repeats until End of Record			

OEF Table of element forces

Also contains composite failure indices and a analysis types (real and complex) and SORT1 and SORT2 formats.

Record 0 -- HEADER

Word	Name	Type	Description
1	NAME(2)	CHAR4	Data block name
3	MONTH	I	
4	DAY	I	
5	YEAR	I	
6	UNDEF(2)	none	

MONPNT2

- Find the relevant element

Word	Name	Type	Description
ELTYPE =32		Unused	
2	UNDEF	none	
ELTYPE =33		Quadrilateral plate element (CQUAD4)	
SCODE,6 =0		Stress	
TCODE,7 =0		Real	
2	FD1	2	FD1
		3	SX1
		4	SY1
		5	TXY1
		6	SA1
		7	SMJRP1
		8	SMNRP1
		9	SMAX1

- Scroll down to the right output type
- Select the required output quantity

MONPNT2

- EID defines the element ID

1	2	3	4	5	6	7	8	9	10
MONPNT2	NAME	LABEL							
	TABLE	TYPE	NDDLiten	EID					

```

MONPNT2  SX1      NORMAL IN X AT Z1
          STRESS   CQUAD4  SX1      2
MONPNT2  SY1      NORMAL IN Y AT Z1
          STRESS   CQUAD4  SY1      2
MONPNT2  TXY1     SHEAR  IN XY AT Z1
          STRESS   CQUAD4  TXY1     2
    
```

- It is reported in the EID column in the output

VALUE	EID	ETYPE	CLASS	RESPONSE	LABEL	SUBCASE NO.					
-8.136726E+04	2	CQUAD4	STRESS	SX1	SX1	NORMAL	IN	X	AT	Z1	1

MONPNT3

MONPNT3 Considerations

- **The MONPNT3 is available for SOLs 101, 103, 108, 109, 111, 112, 144 and 200.**
- **If ELEMSET is blank, no contributions are made from the set of elements attached to the grid.**
- **Fictitious grids or elements do not produce error or warning messages.**

MONPNT3

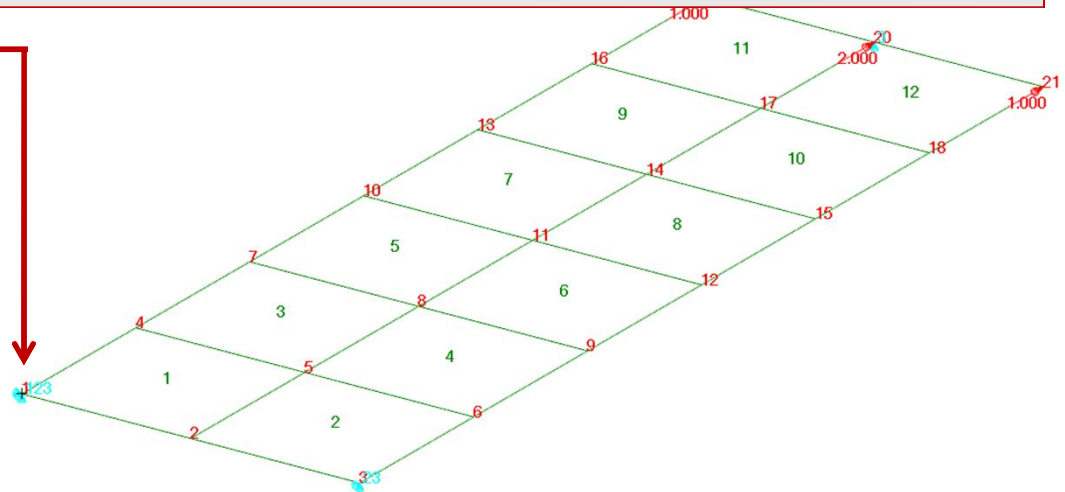
- **MONPNT3 is used to sum free body interface loads**
- **Calculate load contributions at a point from any or all attached entities**
- **Similar to filtered GPFORCE output**
- **Used to create free body diagrams**
- **Resultant forces and moments may be summed about any point**

GPFORCE Primer

- To understand MONPNT3, GPFORCE output is useful

GRID POINT FORCE BALANCE									
POINT-ID	ELEMENT-ID	SOURCE	T1	T2	T3	R1	R2	R3	
1		F-OF-SPC	-6.821210E-13	-2.000000E+03	0.0	0.0	0.0	0.0	
1	1	QUAD4	8.645307E-13	2.000000E+03	0.0	0.0	0.0	0.0	
1		*TOTALS*	1.824096E-13	0.0	0.0	0.0	0.0	0.0	
2	1	QUAD4	8.652083E+02	-2.273737E-13	0.0	0.0	0.0	6.758410E-02	
2	2	QUAD4	-8.652083E+02	1.591616E-12	0.0	0.0	0.0	-6.758410E-02	
2		*TOTALS*	-4.547474E-13	1.364242E-12	0.0	0.0	0.0	0.0	
3		F-OF-SPC	0.0	-2.000000E+03	0.0	0.0	0.0	0.0	
3	2	QUAD4	0.0	2.000000E+03	0.0	0.0	0.0	3.469447E-18	
3		*TOTALS*	0.0	0.0	0.0	0.0	0.0	3.469447E-18	

- Contributions can include
 - Elastic element forces
 - SPCForces
 - MPC/Rigid element forces
 - Applied Loads
 - DMIG & GENEL



GPFORCE Primer

- Standard GPFORCE output
- Contribution from SPC
- Contribution from an element

GRID POINT FORCE BALANCE										
POINT-ID	ELEMENT-ID	SOURCE	T1	T2	T3	R1	R2	R3		
1		F-OFF-SPC	6.821210E-13	-2.000000E+03	0.0	0.0	0.0	0.0		
1	1	QUAD4	8.645307E-13	2.000000E+03	0.0	0.0	0.0	0.0		
1		*TOTALS*	1.824096E-13	0.0	0.0	0.0	0.0	0.0		
2	1	QUAD4	8.652083E+02	-2.273737E-13	0.0	0.0	0.0	6.758410E-02		
2	2	QUAD4	-8.652083E+02	1.591616E-12	0.0	0.0	0.0	-6.758410E-02		
2		*TOTALS*	-4.547474E-13	1.364242E-12	0.0	0.0	0.0	0.0		
3		F-OFF-SPC	0.0	-2.000000E+03	0.0	0.0	0.0	0.0		
3	2	QUAD4	0.0	2.000000E+03	0.0	0.0	0.0	3.469447E-18		
3		*TOTALS*	0.0	0.0	0.0	0.0	0.0	3.469447E-18		
19		APP-LOAD	0.0	1.000000E+03	0.0	0.0	0.0	0.0		
19	11	QUAD4	2.817531E-13	-1.000000E+03	0.0	0.0	0.0	-4.925666E-17		
19		*TOTALS*	2.817531E-13	-6.821210E-13	0.0	0.0	0.0	-4.925666E-17		
20		APP-LOAD	0.0	2.000000E+03	0.0	0.0	0.0	0.0		
20	11	QUAD4	2.364357E-01	-1.000000E+03	0.0	0.0	0.0	-3.612622E-06		
20	12	QUAD4	2.364357E-01	-1.000000E+03	0.0	0.0	0.0	3.612622E-06		
20		*TOTALS*	1.700862E-13	1.591616E-12	0.0	0.0	0.0	5.314285E-17		

- Contribution from applied load
- *TOTALS*** is the summation of all included contributions. This may not be binary zero due to round-off error *OR* may be non-zero due to sources that are not explicitly reported (slide-line forces, superelements, ...)

MONPNT3 - Statics

- Compare MONPNT3 output with standard output

G R I D P O I N T F O R C E B A L A N C E										
POINT-ID	ELEMENT-ID	SOURCE	T1	T2	T3	R1	R2	R3		
11	5 QUAD4	4.507064E+01	-9.984300E+02	0.0	0.0	0.0	2.485460E-04			
11	6 QUAD4	-4.507064E+01	-9.984300E+02	0.0	0.0	0.0	-2.485460E-04			
11	7 QUAD4	8.242488E+00	9.984300E+02	0.0	0.0	0.0	2.485460E-04			
11	8 QUAD4	-8.242488E+00	9.984300E+02	0.0	0.0	0.0	-2.485460E-04			
11	*TOTALS*	1.222134E-12	-1.364242E-12	0.0	0.0	0.0	-9.557242E-17			

S T R U C T U R A L I N T E G R A T E D F R E E B O D Y M O N I T O R P O I N T L O A D S (MONPNT3)										
MONITOR POINT NAME = TYSUM							SUBCASE NO. 1			
LABEL = SUM OF TRANSLATIONAL FORCES IN Y DIRECTION										
CP	=	0	X = 5.00000E-01	Y = 1.50000E+00	Z = 0.00000E+00					
			AXIS	REST. APPLIED						
			----	-----						
			CY	-9.984300E+02						

MONPNT3 - Statics

- The output is obtained by using the MONPNT3 bulk data entry

1	2	3	4	5	6	7	8	9	10
MONPNT3	NAME	LABEL							
	AXES	GRIDSET	ELEMSET	CP	X	Y	Z	XFLAG	

- For example, the output above was obtained using the following MONPNT3 entry

```

MONPNT3  TYSUM  SUM OF TRANSLATIONAL FORCES IN Y DIRECTION
          2      1      2      0      0.5      1.5      0.      SMAD
SET1      1      11
SET1      2      5
    
```


MONPNT3 - Statics

- AXES determines which components are reported

1	2	3	4	5	6	7	8	9	10
MONPNT3	NAME	LABEL							
	AXES	GRIDSET	ELEMSET	CP	X	Y	Z	XFLAG	

```

MONPNT3  TYSUM  SUM OF TRANSLATIONAL FORCES IN Y DIRECTION
          2      1      2      0      0.5      1.5      0.      SMAD
SET1      1      11
SET1      2      5
  
```

STRUCTURAL INTEGRATED FREE BODY MONITOR POINT LOADS
(MONPNT3)

MONITOR POINT NAME = TYSUM SUBCASE NO. 1
 LABEL = SUM OF TRANSLATIONAL FORCES IN Y DIRECTION
 CP = 0 X = 5.00000E-01 Y = 1.50000E+00 Z = 0.00000E+00

AXIS	REST. APPLIED
----	-----
CY	-9.984300E+02

MONPNT3 - Statics

- GRIDSET** defines which GRID points are used in the summation

1	2	3	4	5	6	7	8	9	10
MONPNT3	NAME	LABEL							
	AXES	GRIDSET	ELEMSET	CP	X	Y	Z	XFLAG	

```

MONPNT3  TYSUM  SUM OF TRANSLATIONAL FORCES IN Y DIRECTION
          2      1      2      0      0.5      1.5      0.      SMAD
SET1      1      11
SET1      2      5
  
```

- Here only 1 GRID point appears on the SET1 entry, but any number of GRID points may be defined.
 - For example, the total load transmitted in a bolt group may be calculated

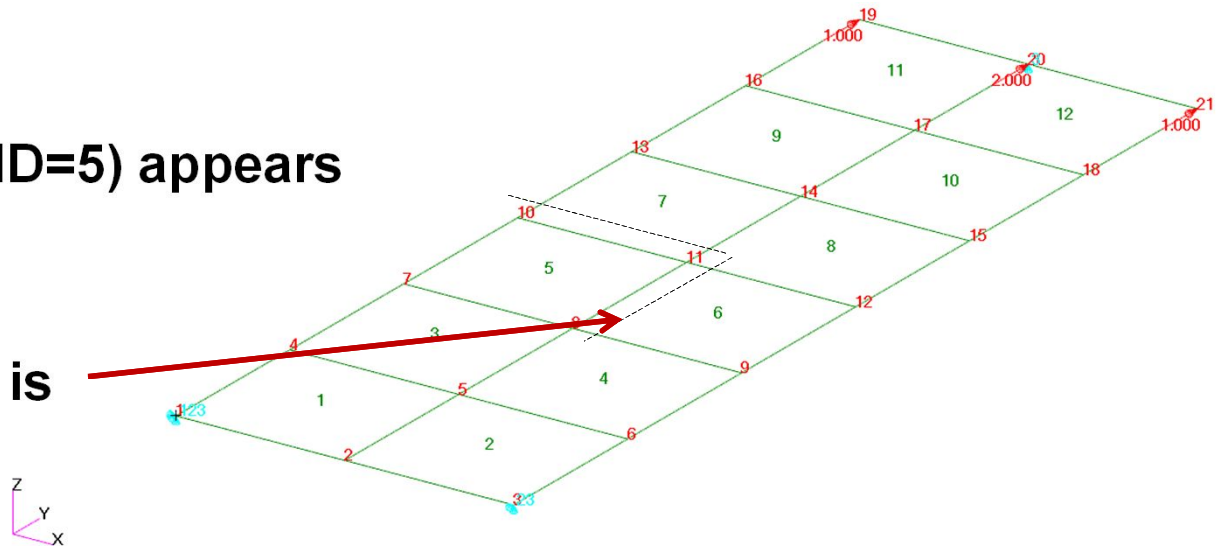
MONPNT3 - Statics

- ELEMSET defines which elements are used for the “cut”

1	2	3	4	5	6	7	8	9	10
MONPNT3	NAME	LABEL							
	AXES	GRIDSET	ELEMSET	CP	X	Y	Z	XFLAG	

MONPNT3	TYSUM	SUM OF TRANSLATIONAL FORCES IN Y DIRECTION					SMAD
	2	1	2	0	0.5	1.5	0.
SET1	1	11					
SET1	2	5					

- Here only 1 element (ID=5) appears on the SET1 list
- This is like the model is “cut” here



MONPNT3 - Statics

- CP is the coordinate system in which X,Y,Z are defined

1	2	3	4	5	6	7	8	9	10
MONPNT3	NAME	LABEL							
	AXES	GRIDSET	ELEMSET	CP	X	Y	Z	XFLAG	

MONPNT3	TYSUM	SUM OF TRANSLATIONAL FORCES IN Y DIRECTION						
	2	1	2	0	0.5	1.5	0.	SMAD
SET1	1	11						
SET1	2	5						

STRUCTURAL INTEGRATED FREE BODY MONITOR POINT LOADS
(MONPNT3)

MONITOR POINT NAME = TYSUM SUBCASE NO. 1
 LABEL = SUM OF TRANSLATIONAL FORCES IN Y DIRECTION
 CP = 0 X = 5.00000E-01 Y = 1.50000E+00 Z = 0.00000E+00

AXIS	REST. APPLIED
----	-----
CY	-9.984300E+02

MONPNT3 - Statics

- X,Y,Z coordinates (in CP) of the summation location

1	2	3	4	5	6	7	8	9	10
MONPNT3	NAME	LABEL							
	AXES	GRIDSET	ELEMSET	CP	X	Y	Z	XFLAG	

MONPNT3	TYSUM	SUM OF TRANSLATIONAL FORCES IN Y DIRECTION						
	2	1	2	0	0.5	1.5	0.	SMAD
SET1	1	11						
SET1	2	5						

STRUCTURAL INTEGRATED FREE BODY MONITOR POINT LOADS
(MONPNT3)

MONITOR POINT NAME = TYSUM SUBCASE NO. 1

LABEL = SUM OF TRANSLATIONAL FORCES IN Y DIRECTION

CP = 0 X = 5.00000E-01 Y = 1.50000E+00 Z = 0.00000E+00

AXIS REST. APPLIED

CY -9.984300E+02

MONPNT3 - Statics

- **XFLAG** to exclude contributions from selected sources

1	2	3	4	5	6	7	8	9	10
MONPNT3	NAME	LABEL							
	AXES	GRIDSET	ELEMSET	CP	X	Y	Z	XFLAG	

MONPNT3	TYSUM	SUM OF TRANSLATIONAL FORCES IN Y DIRECTION						
	2	1	2	0	0.5	1.5	0.	SMAD
SET1	1	11						
SET1	2	5						

- **Possible sources are**
 - S - Forces of single point constraint
 - M - Forces of multi-point constraint
 - A, L or P - Applied loads
 - D - DMIG and other sources

MONPNT3 - Statics

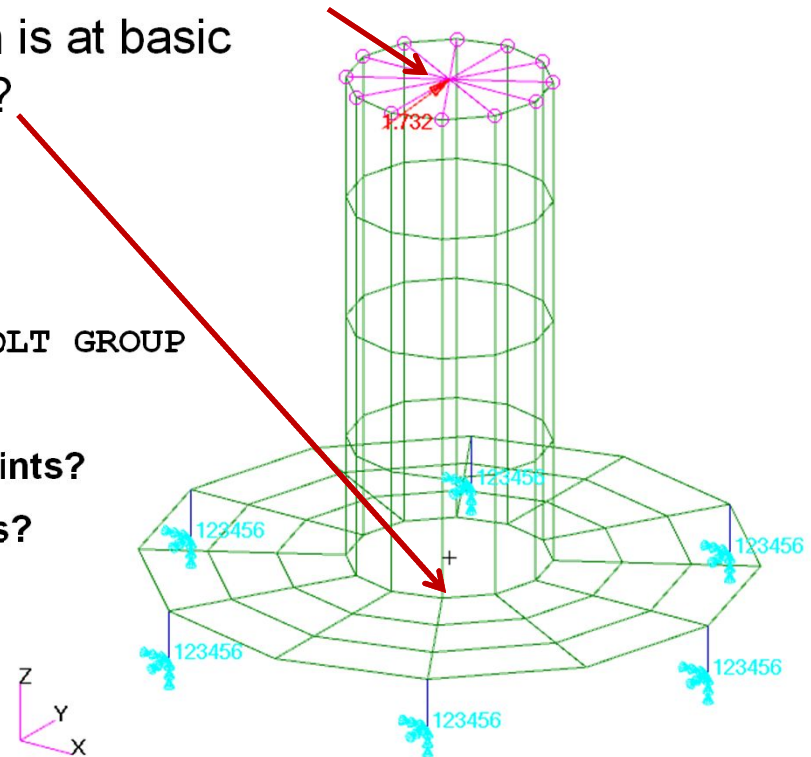
- Now we can sum forces about any point, over any domain of GRID points, for any contributing elements
- Calculate the total load of the bolt group attaching a flange
 - What are the total shear forces and moments due to this force about the centre of the bolt group, which is at basic (0.0,0.0,-0.5); that's approximately here?

- Easy – use MONPNT3

```
MONPNT3  VMT      TOTAL SHEAR/MOMENT FOR BOLT GROUP
,123456,1,2,0,0.,0.,-.5,SMAD
SET1,1,4,12,24,36,48,60
SET1,2,85,THRU,90
```

← Which GRID points?

← Which elements?



MONPNT3 - Statics

- Automatic summation about any point

STRUCTURAL INTEGRATED FREE BODY MONITOR POINT LOADS (MONPNT3)

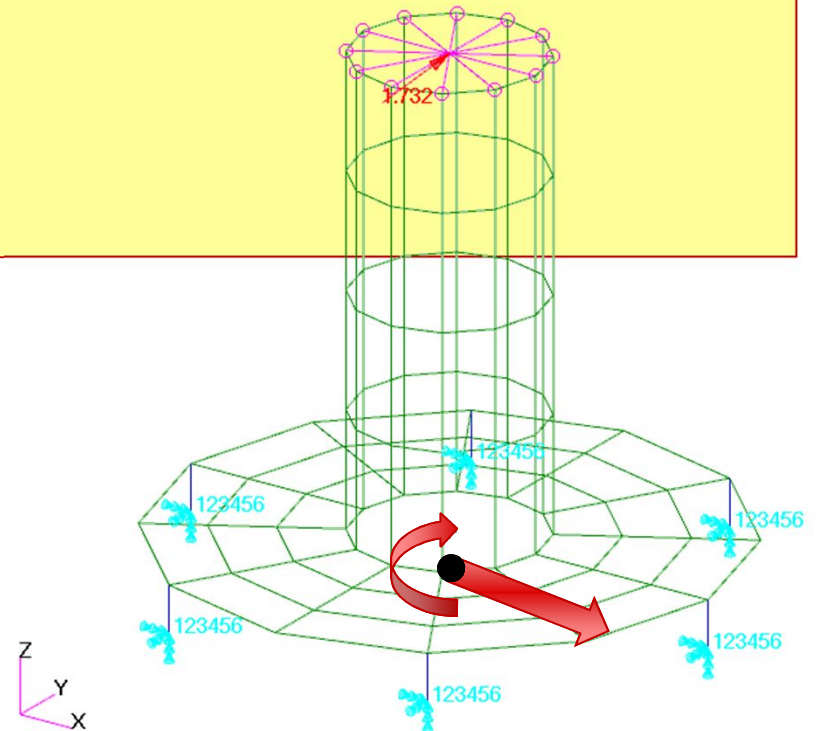
MONITOR POINT NAME = VMT

SUBCASE NO. 1

LABEL = TOTAL SHEAR/MOMENT FOR BOLT GROUP

CP = 0 X = 0.00000E+00 Y = 0.00000E+00 Z = -5.00000E-01

AXIS	REST. APPLIED
----	-----
CX	-1.000000E+00
CY	-1.000000E+00
CZ	-1.000000E+00
CMX	5.500000E+00
CMY	-5.500000E+00
CMZ	2.201835E-10



MONPNT3 - Dynamics

- The same MONPNT3 entry can used in dynamic analysis

1	2	3	4	5	6	7	8	9	10
MONPNT3	NAME	LABEL							
	AXES	GRIDSET	ELEMSET	CP	X	Y	Z	XFLAG	

```

MONPNT3  VMT      TOTAL SHEAR/MOMENT FOR BOLT GROUP
,123456,1,2,0,0.,0.,-.5,SMAD
SET1,1,4,12,24,36,48,60
SET1,2,85,THRU,90
    
```

- For example, apply the load in a frequency response analysis

MONPNT3 - Dynamics

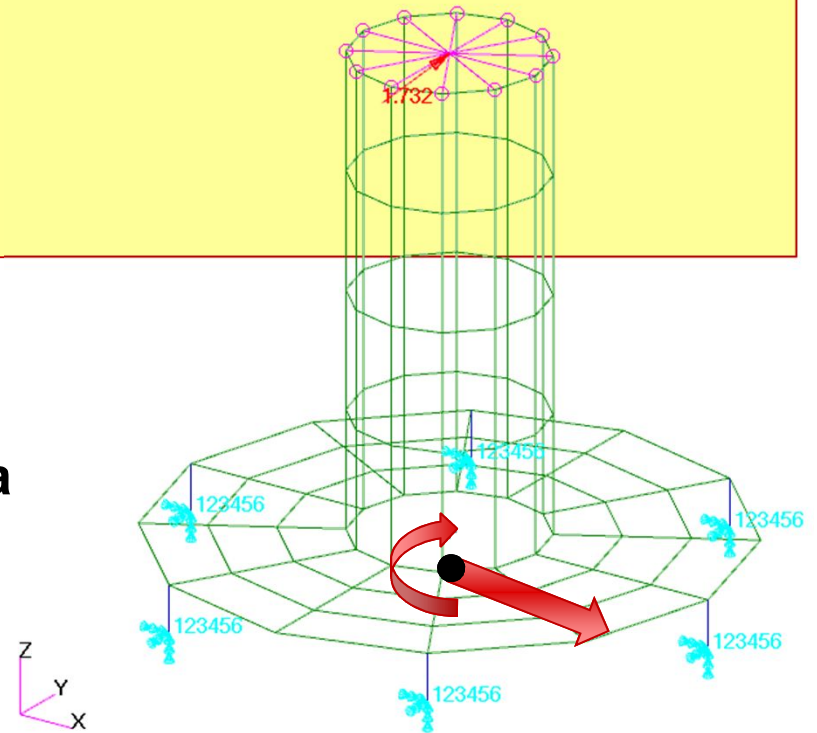
- Resultant output from frequency response analysis

STRUCTURAL INTEGRATED FREE BODY MONITOR POINT LOADS (MONPNT3)
(REAL/IMAGINARY)

MONITOR POINT NAME = VMT COMPONENT = CX SUBCASE NO. 1
LABEL = TOTAL SHEAR/MOMENT FOR BOLT GROUP
CP = 0 X = 0.000000E+00 Y = 0.000000E+00 Z = -5.000000E-01

FREQUENCY	RESULTANT
2.000000E+01	4.105549E-01 -2.590404E-03

- Notice the output is complex
(Real/Imaginary in this case), and is a
function of frequency and direction



MONDSP1

MONDSP1 Considerations

- **The MONDSP1 is available for SOLs 101, 103, 105, 108, 109, 111, 112, 144, 146 and 200.**
- **The entry can create either an aerodynamic or a structural MONDSP1, depending on the data provided in the COMP field.**
 - A structural MONDSP1 can have the same name as an aerodynamic MONDSP1, but two structural or two aerodynamic MONDSP1s cannot have the same name.
- **The INDDOF field defines the Ci field on the virtual RBE3; that is, it defines the components of the grids on the AECOMP that will be sampled to define the dependent (monitor point) displacement.**
 - Typically, the default is the correct choice. However, if there is only a single grid point, all six DOFs can be used.

MONDSP1

- **MONDSP1 calculates the average displacement from a collection of GRID points**
- **It behaves in exactly the same way as a uniformly weighted RBE3**
- **However, no RBE3 element is required and no additional GRID points are required**
- **The output may be in any required coordinate system**

MONDSP1 - Statics

- Compare MONDSP1 output with standard output from an RBE3

```
DISPLACEMENT VECTOR
POINT ID. TYPE      T1      T2      T3      R1      R2      R3
1001      G  1.659948E-09  1.659947E-09  6.513186E-06 -1.007604E-05  1.007602E-05 -2.122635E-16
```

```
STRUCTURAL MONITOR POINT DISPLACEMENTS
MONITOR POINT NAME = AVGSTN1      COMPONENT = 30      CLASS = GENERAL      SUBCASE NO. 1
LABEL = AVERAGE DISPLACEMENT STATION 1
CP = 0      X = 0.00000E+00      Y = 0.00000E+00      Z = 0.00000E+00      CD = 0

AXIS      ELASTIC REST.
----      -
CX      1.659947E-09
CY      1.659947E-09
CZ      6.513186E-06
CMX      -1.007604E-05
CMY      1.007602E-05
CMZ      -2.122077E-16
```

MONDSP1 - Statics

- The output is obtained by using the MONDSP1 bulk data entry

1	2	3	4	5	6	7	8	9	10
MONDSP1	NAME	LABEL							
	AXES	COMP	CP	X	Y	Z	CD	INDDOF	

- For example, the output above was obtained using the following MONDSP1 entry

```

MONDSP1  AVGSTN1  AVERAGE  DISPLACEMENT  STATION 1
          123456  30        0          0.0        0.0        0.0        0        123
SET1      30        1        5          9        17        21        29        33
          41        45        53         57         65
    
```

MONDSP1 - Statics

- AXES determines which components are reported

1	2	3	4	5	6	7	8	9	10
MONDSP1	NAME	LABEL							
	AXES	COMP	CP	X	Y	Z	CD	INDDOF	

```

MONDSP1  AVGSTN1  AVERAGE DISPLACEMENT STATION 1
          123456  30      0      0.0      0.0      0.0      0      123
SET1      30      1      5      9      17      21      29      33
          41      45      53      57      65
  
```

STRUCTURAL MONITOR POINT DISPLACEMENTS

```

MONITOR POINT NAME = AVGSTN1      COMPONENT = 30      CLASS = GENERAL      SUBCASE NO. 1
LABEL = AVERAGE DISPLACEMENT STATION 1
CP = 0      X = 0.00000E+00      Y = 0.00000E+00      Z = 0.00000E+00      CD = 0
  
```

AXIS	ELASTIC REST.
CX	1.659947E-09
CY	1.659947E-09
CZ	6.513186E-06
CMX	-1.007604E-05
CMY	1.007602E-05
CMZ	-2.122077E-16

MONDSP1 - Statics

- COMP selects the SET1 to define the GRID points for averaging

1	2	3	4	5	6	7	8	9	10
MONDSP1	NAME	LABEL							
	AXES	COMP	CP	X	Y	Z	CD	INDDOF	

```

MONDSP1  AVGSTN1  AVERAGE DISPLACEMENT STATION 1
          123456  30      0      0.0      0.0      0.0      0      123
SET1      30      1      5      9      17      21      29      33
          41      45      53      57      65
    
```

```

S T R U C T U R A L   M O N I T O R   P O I N T   D I S P L A C E M E N T S

MONITOR POINT NAME = AVGSTN1      COMPONENT = 30      CLASS = GENERAL      SUBCASE NO. 1
LABEL = AVERAGE DISPLACEMENT STATION 1
CP = 0      X = 0.00000E+00      Y = 0.00000E+00      Z = 0.00000E+00      CD = 0

  AXIS      ELASTIC REST.
  ----      -
  CX      1.659947E-09
  CY      1.659947E-09
  CZ      6.513186E-06
  CMX     -1.007604E-05
  CMY      1.007602E-05
  CMZ     -2.122077E-16
    
```

MONDSP1 - Statics

- CP is the coordinate system in which X,Y,Z are defined

1	2	3	4	5	6	7	8	9	10
MONDSP1	NAME	LABEL							
	AXES	COMP	CP	X	Y	Z	CD	INDDOF	

```

MONDSP1  AVGSTN1  AVERAGE DISPLACEMENT STATION 1
          123456  30      0      0.0      0.0      0.0      0      123
SET1      30      1      5      9      17      21      29      33
          41      45      53      57      65
    
```

```

S T R U C T U R A L   M O N I T O R   P O I N T   D I S P L A C E M E N T S

MONITOR POINT NAME = AVGSTN1      COMPONENT = 30      CLASS = GENERAL      SUBCASE NO. 1
LABEL = AVERAGE DISPLACEMENT STATION 1
CP = 0      X = 0.00000E+00      Y = 0.00000E+00      Z = 0.00000E+00      CD = 0

  AXIS      ELASTIC REST.
  ----      -
  CX      1.659947E-09
  CY      1.659947E-09
  CZ      6.513186E-06
  CMX     -1.007604E-05
  CMY      1.007602E-05
  CMZ     -2.122077E-16
    
```

MONDSP1 - Statics

- X,Y,Z coordinates (in CP) of the averaging location

1	2	3	4	5	6	7	8	9	10
MONDSP1	NAME	LABEL							
	AXES	COMP	CP	X	Y	Z	CD	INDDOF	

```

MONDSP1  AVGSTN1  AVERAGE DISPLACEMENT STATION 1
          123456  30      0      0.0      0.0      0.0      0      123
SET1      30      1      5      9      17      21      29      33
          41      45      53      57      65
  
```

```

S T R U C T U R A L   M O N I T O R   P O I N T   D I S P L A C E M E N T S

MONITOR POINT NAME = AVGSTN1      COMPONENT = 30      CLASS = GENERAL      SUBCASE NO. 1
LABEL = AVERAGE DISPLACEMENT STATION 1
CP = 0      X = 0.00000E+00      Y = 0.00000E+00      Z = 0.00000E+00      CD = 0

  AXIS      ELASTIC REST.
  ----      -
  CX      1.659947E-09
  CY      1.659947E-09
  CZ      6.513186E-06
  CMX     -1.007604E-05
  CMY      1.007602E-05
  CMZ     -2.122077E-16
  
```

MONDSP1 - Statics

- CD defines the output coordinate system

1	2	3	4	5	6	7	8	9	10
MONDSP1	NAME	LABEL							
	AXES	COMP	CP	X	Y	Z	CD	INDDOF	

```

MONDSP1  AVGSTN1  AVERAGE DISPLACEMENT STATION 1
          123456  30      0      0.0      0.0      0.0      0      123
SET1      30      1      5      9      17      21      29      33
          41      45      53      57      65
    
```

```

S T R U C T U R A L   M O N I T O R   P O I N T   D I S P L A C E M E N T S

MONITOR POINT NAME = AVGSTN1      COMPONENT = 30      CLASS = GENERAL      SUBCASE NO. 1
LABEL = AVERAGE DISPLACEMENT STATION 1
CP = 0      X = 0.00000E+00      Y = 0.00000E+00      Z = 0.00000E+00      CD = 0

  AXIS  ELASTIC REST.
  ----  -
  CX    1.659947E-09
  CY    1.659947E-09
  CZ    6.513186E-06
  CMX   -1.007604E-05
  CMY    1.007602E-05
  CMZ   -2.122077E-16
    
```

MONDSP1 - Statics

- INDDOF** - components from which the displacements are derived

1	2	3	4	5	6	7	8	9	10
MONDSP1	NAME	LABEL							
	AXES	COMP	CP	X	Y	Z	CD	INDDOF	

MONDSP1	AVGSTN1	AVERAGE	DISPLACEMENT	STATION 1					
	123456	30	0	0.0	0.0	0.0	0	123	
SET1	30	1	5	9	17	21	29	33	
	41	45	53	57	65				

```

S T R U C T U R A L   M O N I T O R   P O I N T   D I S P L A C E M E N T S

MONITOR POINT NAME = AVGSTN1      COMPONENT = 30      CLASS = GENERAL      SUBCASE NO. 1
LABEL = AVERAGE DISPLACEMENT STATION 1
CP = 0      X = 0.00000E+00      Y = 0.00000E+00      Z = 0.00000E+00      CD = 0

  AXIS    ELASTIC REST.
  ----    -
  CX      1.659947E-09
  CY      1.659947E-09
  CZ      6.513186E-06
  CMX     -1.007604E-05
  CMY      1.007602E-05
  CMZ     -2.122077E-16
    
```

MONDSP1 - Statics

- Compare the MONDSP1 & RBE3 input

1	2	3	4	5	6	7	8	9	10
MONDSP1	NAME	LABEL							
	AXES	COMP	CP	X	Y	Z	CD	INDDOF	

1	2	3	4	5	6	7	8	9	10
RBE3	EID		REFGRID	REFC	WT1	C1	G1,1	G1,2	
	G1,3	WT2	C2	G2,1	G2,2	-etc.-	WT3	C3	
	G3,1	G3,2	-etc.-	WT4	C4	G4,1	G4,2	-etc.-	
	“UM”	GM1	CM1	GM2	CM2	GM3	CM3		
		GM4	CM4	GM5	CM5	-etc.-			
	“ALPHA”	ALPHA							

- Are you comfortable coding these fields?
- What happens if REFGRID is already dependent on something else?
- If the RBE3 frightens you, the “UM” option will give you nightmares

Combining Monitor Point Outputs

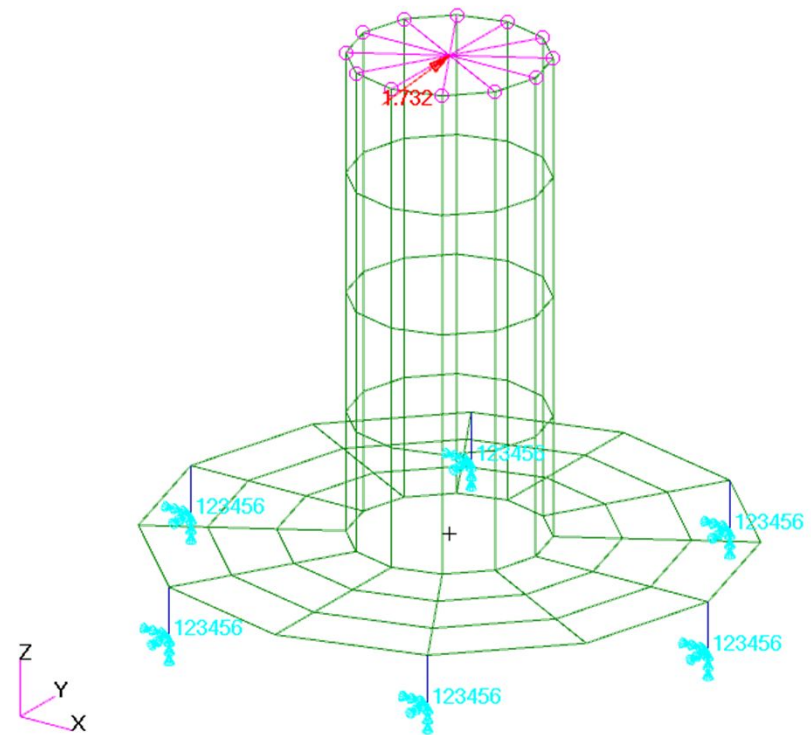
MONSUM

- **MONSUM** may be used to output a weighted sum of existing monitor results
- **Both aero and structural Monitor Points may be defined as input**
 - We will concentrate on the structural inputs for use in dynamic analysis
- **Only some of the Monitor Point types may be used as input to a MONSUM**
 - MONPNT1
 - MONPNT3
 - MONDSP1
- **MONSUM requires the Monitor Point input types to be similar**
 - Force and moment summation monitor points: MONPNT1 & MONPNT3
 - Average displacement monitor points: MONDSP1
 - It is not possible to MONSUM MONPNT1 & MONDSP1, for example

MONSUM - Example 1

- Let's assume we want to calculate the average axial force in the 6 BUSH elements attaching this flange
- MONPNT3 will be used to obtain the BUSH axial forces

```
MONPNT3,BF85,BUSH LOAD 85  
,3,851,852,,3.,,SMAD  
SET1,851,4  
SET1,852,85
```



MONSUM - Example 1

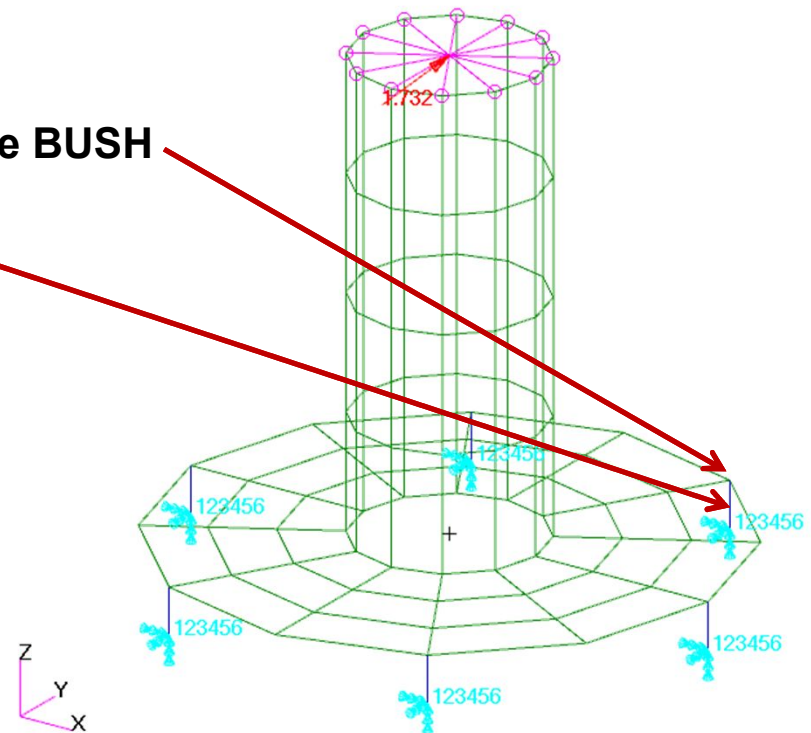
- Let's assume we want to calculate the average axial force in the 6 BUSH elements attaching this flange
- MONPNT3 will be used to obtain the BUSH axial forces

Axial force **Coordinates of GRID point in GRIDSET**

```
MONPNT3,BF85,BUSH LOAD 85  
,3,851,852,,3.,0.,0.,SMAD  
SET1,851,4 GRID point at the end of the BUSH  
SET1,852,85 BUSH element ID
```

- Repeat for all 6 bushes

```
MONPNT3,BF86,BUSH LOAD 86  
,3,861,862,,1.5,2.598076,,SMAD  
SET1,861,12  
SET1,862,86  
...
```



MONSUM - Example 1

- To get all 6 Monitor Points defined

MONPNT3,BF85,BUSH LOAD 85

,3,851,852,,3.,,,SMAD

SET1,851,4

SET1,852,85

MONPNT3,BF86,BUSH LOAD 86

,3,861,862,,1.5,2.598076,,SMAD

SET1,861,12

SET1,862,86

MONPNT3,BF87,BUSH LOAD 87

,3,871,872,,-1.5,2.598076,,SMAD

SET1,871,24

SET1,872,87

MONPNT3,BF88,BUSH LOAD 88

,3,881,882,,-3.0,,SMAD

SET1,881,36

SET1,882,88

MONPNT3,BF89,BUSH LOAD 89

,3,891,892,,-1.5,-2.59808,,SMAD

SET1,891,48

SET1,892,89

MONPNT3,BF90,BUSH LOAD 90

,3,901,902,,1.5,-2.59808,,SMAD

SET1,901,60

SET1,902,90

MONSUM - Example 1

- Define the MONSUM entry to calculate the average force

1	2	3	4	5	6	7	8	9	10
MONSUM	NAME	LABEL							
	MTYPE	NEWAXISA	NAME1A	AXES1A	COEF1A	NAME2A	AXES2A	COEF2A	
			NAME3A	AXES3A	COEF3A	etc.			
		NEWAXISB	NAME2A	AXES1B	COEF1B	NAME2B	AXIS2B	COEF2B	
			NAME3B	AXES3B	COEF3B	etc.			
		NEWAXISC	etc.						

- MTYPE is set to MONPNT3
- NEWAXISA can be set to any of the numbers 1 to 6 – let's use 1 to store the average force in the "CX" place
- Then list the names of the MONPNT3, component and coefficient values in fields NAMEiA, AXESiA, & COEFiA, respectively

MONSUM - Example 1

- The completed MONSUM entry

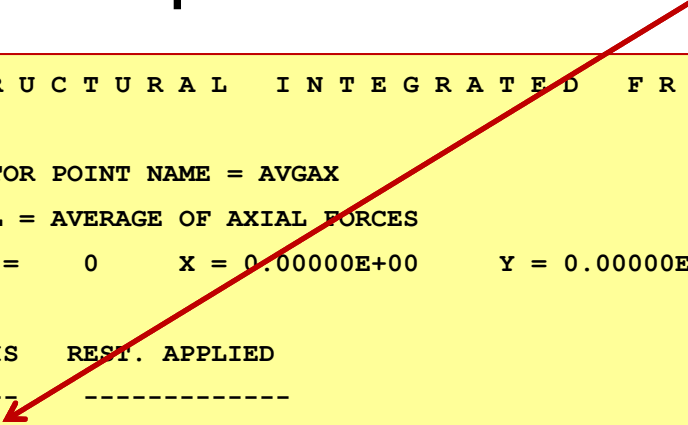
MONSUM	AVGAX	AVERAGE OF AXIAL FORCES					
	MONPNT3 1	BF85	3	.1666667	BF86	3	.1666667
		BF87	3	.1666667	BF88	3	.1666667
		BF89	3	.1666667	BF90	3	.1666667

- The output is stored in the CX component, as requested

```
STRUCTURAL INTEGRATED FREE BODY MONITOR POINT LOADS (MONPNT3)

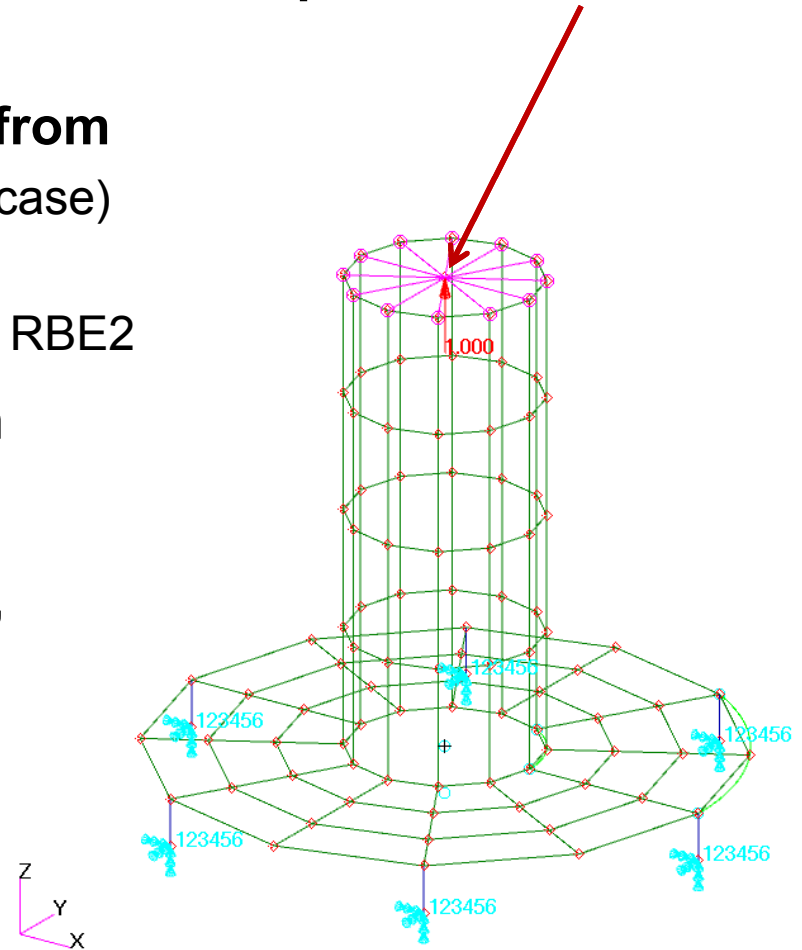
MONITOR POINT NAME = AVGAX                      SUBCASE NO. 1
LABEL = AVERAGE OF AXIAL FORCES
CP = 0      X = 0.000000E+00      Y = 0.000000E+00      Z = 0.000000E+00

AXIS  REST. APPLIED
-----
CX    -1.666667E-01
```



MONSUM - Example 2

- Let's add a lump mass (1000.0 units) to the GRID point at the centre of the RBE2
- We want to see the load contribution from
 - The applied load (Z direction only in this case)
 - The lump mass inertial load
 - The inertial load from the structure at the RBE2
- We also want to check for equilibrium
- To do this, we need to use MONPNT1, MONPNT3 and MONSUM



MONSUM - Example 2

- Here are the Monitor Point entries

```
MONPNT1,ZMASS,ZLOAD MASS  
,3,30,,0.,0.,5.
```

```
SET1,30,117
```

```
MONPNT1,ZRING,ZLOAD RING  
,3,31,,0.,0.,5.
```

```
SET1,31,105,THRU,116
```

```
MONPNT3,ZRBE2,ZLOAD RBE2  
,3,31,32,,0.,0.,5.
```

```
SET1,32,9191
```

```
$
```

```
MONSUM,DIFF,DIFF IN INERTIAL LOADS  
,1,MONPNT3,ZRBE2,3,1.
```

```
,,SMONPNT1,ZMASS,3,-1.,ZRING,3,-1.
```

 Report inertial load and applied load at the lump mass point

 Report inertial load and applied load at the end GRID points

 Report the resultant load (end GRID points & lump mass & applied load)

 Add up the loads and check for equilibrium

MONSUM - Example 2

- Lump mass = 1000.0

COMPLEX ACCELERATION VECTOR (REAL/IMAGINARY)

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
117	G	-1.862493E-16	-2.720331E-15	2.700518E-04	1.712347E-17	-3.337248E-17	-2.736424E-14
		0.0	0.0	0.0	0.0	0.0	0.0

STRUCTURAL MONITOR POINT INTEGRATED LOADS (MONPNT1) (REAL/IMAGINARY)

MONITOR POINT NAME = ZMASS COMPONENT = CZ GENERAL SUBCASE NO. 1
 LABEL = ZLOAD MASS
 CP = 0 X = 0.000000E+00 Y = 0.000000E+00 Z = 5.000000E+00 CD = 0

FREQUENCY	INERTIAL	EXTERNAL	FLEXIBLE INCREMENT	GUST	TOTAL AERO	TOTAL
2.000000E+01	-2.700518E-01	1.000000E+00			0.000000E+00	7.299482E-01
	0.000000E+00	0.000000E+00			0.000000E+00	0.000000E+00

MONSUM - Example 2

- Results

STRUCTURAL MONITOR POINT INTEGRATED LOADS (MONPNT1)
(REAL/IMAGINARY)

MONITOR POINT NAME = ZRING COMPONENT = CZ GENERAL SUBCASE NO. 1
LABEL = ZLOAD RING
CP = 0 X = 0.000000E+00 Y = 0.000000E+00 Z = 5.000000E+00 CD = 0

FREQUENCY	INERTIAL	EXTERNAL	FLEXIBLE INCREMENT	GUST	TOTAL AERO	TOTAL
2.000000E+01	-8.387344E-02	0.000000E+00			0.000000E+00	-8.387344E-02
	0.000000E+00	0.000000E+00			0.000000E+00	0.000000E+00

STRUCTURAL INTEGRATED FREE BODY MONITOR POINT LOADS (MONPNT3)
(REAL/IMAGINARY)

MONITOR POINT NAME = ZRBE2 COMPONENT = CZ SUBCASE NO. 1
LABEL = ZLOAD RBE2
CP = 0 X = 0.000000E+00 Y = 0.000000E+00 Z = 5.000000E+00

FREQUENCY	RESULTANT
2.000000E+01	6.460748E-01
	0.000000E+00

MONSUM - Example 2

- Results

STRUCTURAL MONITOR POINT INTEGRATED LOADS (MONPNT1)
(REAL/IMAGINARY)

MONITOR POINT NAME = DIFF COMPONENT = CX GENERAL SUBCASE NO. 1
LABEL =
CP = 0 X = 0.000000E+00 Y = 0.000000E+00 Z = 0.000000E+00 CD = 0

FREQUENCY	INERTIAL	EXTERNAL	FLEXIBLE INCREMENT	GUST	TOTAL AERO	TOTAL
2.000000E+01	3.539252E-01 0.000000E+00	-3.539252E-01 0.000000E+00			0.000000E+00 0.000000E+00	5.960464E-08 0.000000E+00

- $6.460748E-01 - (7.299482E-01 + (-8.387344E-02)) = 4.0E-08$

MONITOR

(Case Control)

MONITOR

- The **MONITOR** case control command is required to switch on the output of Monitor Points

$$\text{MONITOR}[\text{REAL or IMAG, PHASE, NODSP1, NOPNT1, NOPNT2, NOPNT3}] = \left\{ \begin{array}{l} \text{ALL} \\ \text{NONE} \end{array} \right\}$$

- For frequency response, it may be used to obtain the complex output in **REAL/IMAGINARY** or **MAGNITUDE PHASE** format
- It may be used to only output some of the monitor point types
- It may not be used to define a **SET** of monitor entries; i.e. only **ALL** or **NONE** is supported
- Other data recovery requests are not required

WORKSHOP 9

Example of Monitor Points



SECTION 17

Adams MNF

FLEX BODY FOR ADAMS

- **Adams is primary used for rigid body mechanical system simulation (MSS)**
- **Selective components of this rigid body system can be replaced with a flexible body for more accurate durability analysis**
- **Accomplished by generating Craig-Bampton modes using MNF file in MSC Nastran**
 - This MNF file is then read into Adams
 - Replace selective rigid body with flexible body
- **Adams MNF file has been supported since v2004+ using SOL 103**

FLEX BODY FOR ADAMS

- **Adams MNF file is now also supported using SOL 400 in 2013+**
 - Allows the inclusion of nonlinear effects (large displacement, material nonlinearity, etc.)
 - Some of these features were available with SOL 106 to 103 restart (2 jobs)
 - Pre-loaded modes may be quite different than the undeformed modes
 - Dynamic simulation conducted around this deformed configuration are still assumed to be small in Adams



ADAMSMNF

SOL 103

REQUIRED INPUT

- The following basic input are required to generate the Adamsmnf file.
 - Additional Options will be discussed later on

SOL 103

CEND

ADAMSMNF FLEXBODY=YES

METHOD = 100

.

Begin Bulk

EIGRL,100,...

DTI,units,1,mass_unit,force_unit,length_unit,time_unit

ASET1,...

QSET1,0,...

SPOINT,....

ADAMSMNF – MINIMUM REQUIREMENT

Requests generation of the ADAMSMNF file

ADAMSMNF FLEXBODY=YES

UNITS

- Units must be specified when using the ADAMSMNF feature

DTI,units,1,mass_unit,force_unit,length_unit,time_unit

Example

dti,units,1,kg,n,m,s

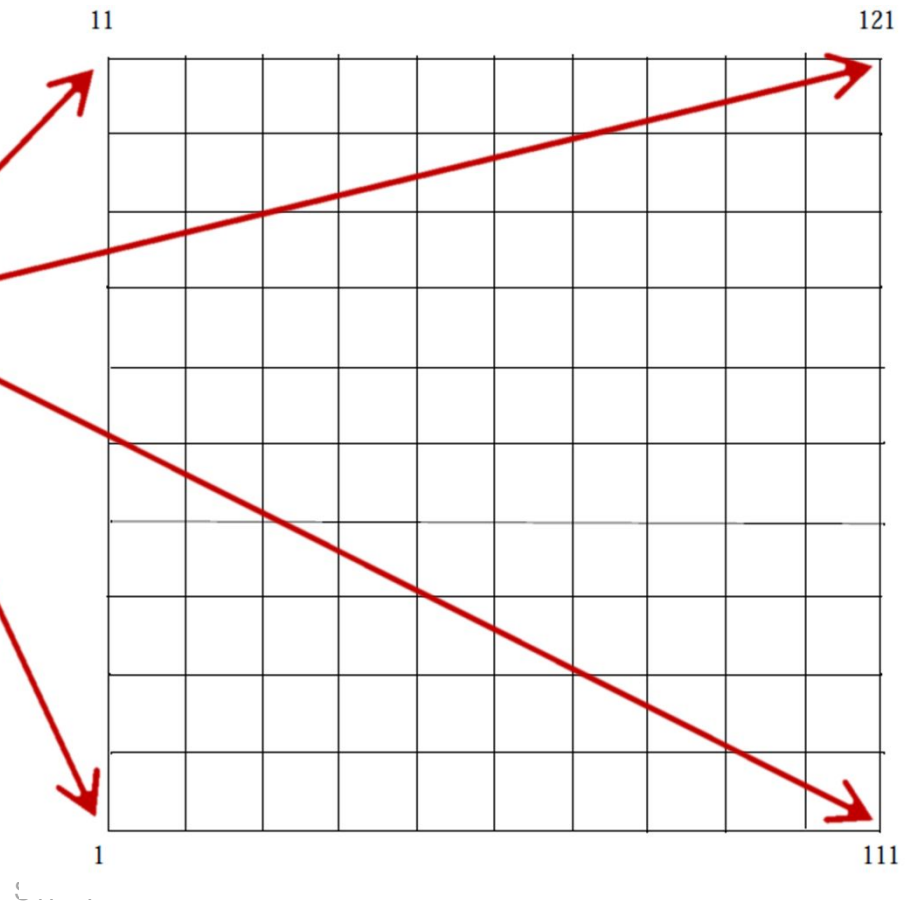
ADAMSMNF – MINIMUM REQUIREMENT (CONT.)

Attachment Points and Component Modes

- Attachment points for Adams are defined with the ASET/ASETi entries

Example

ASET1,123456,1,11,111,121
SPOINT,80001,THRU,80030
QSET1,0,80001,THRU,80030



ADAMSMNF OPTIONS

- All ADAMSMNF Options (See QRG for details)

$$\begin{aligned}
 \text{ADAMSMNF} & \left[\text{FLEXBODY} = \begin{Bmatrix} \text{NO} \\ \text{YES} \end{Bmatrix}, \left[\text{FLEXONLY} = \begin{Bmatrix} \text{YES} \\ \text{NO} \end{Bmatrix} \right], \right. \\
 & \left[\text{ADMCHK} = \begin{Bmatrix} \text{NO} \\ \text{YES} \end{Bmatrix}, \left[\text{ADMOUT} = \begin{Bmatrix} \text{NO} \\ \text{YES} \end{Bmatrix} \right], \right. \\
 & \left[\text{OUTGSTRS} = \begin{Bmatrix} \text{YES} \\ \text{NO} \end{Bmatrix}, \left[\text{OUTGSTRN} = \begin{Bmatrix} \text{YES} \\ \text{NO} \end{Bmatrix} \right], \right. \\
 & \left[\text{OUTSTRS} = \begin{Bmatrix} \text{NO} \\ \text{YES} \end{Bmatrix}, \left[\text{OUTSTRN} = \begin{Bmatrix} \text{NO} \\ \text{YES} \end{Bmatrix} \right] \right. \\
 & \left[\text{V1ORTHO} = \begin{Bmatrix} -1.0 \\ \text{value1} \end{Bmatrix}, \left[\text{V2ORTHO} = \begin{Bmatrix} 1.0\text{e}8 \\ \text{value2} \end{Bmatrix} \right], \right. \\
 & \left[\text{MINVAR} = \begin{Bmatrix} \text{PARTIAL} \\ \text{CONSTANT} \\ \text{FULL} \\ \text{NONE} \\ \text{RIGID} \end{Bmatrix}, \left[\text{PSETID} = \begin{Bmatrix} \text{NONE} \\ \text{setid}_{\text{plotel}} \\ \text{ALL} \end{Bmatrix} \right], \left[\text{EXPORT} = \begin{Bmatrix} \text{MNF} \\ \text{DB} \\ \text{BOTH} \end{Bmatrix} \right] \right]
 \end{aligned}$$

ADAMSMNF OPTIONS (CONT.)

- **FLEXBODY** Requests ADAMSMNF file
- **FLEXONLY** Requests standard MSC Nastran solution and data recovery following MSC Nastran/Adams interface execution
- **ADMCHECK** Requests ADAMSMNF diagnostic messages
- **ADMOUT** Requests generation of op2 file
- **OUTGSTRS** Controls grid point stress output to .op2 file, mnf file, or both
- **OUTGSTRN** Controls grid point strain output to .op2 file, mnf file, or both
- **OUTSTRS** Controls element stress output to .op2 file
- **OUTSTRN** Controls element strain output to .op2 file

ADAMSMNF OPTIONS (CONT.)

- **V1ORTHO** Controls the lowest frequency (hz) for the Craig-Bampton modes
- **V2ORTHO** Controls the highest frequency (hz) for the Craig-Bampton modes
- **MINVAR** Requests the type of mass invariant output
- **EXPORT** Controls modal output to MNF file, MSC Nastran database, or both
- **PSETID** Selects a set of elements (including PLOTEL) whose grids are retained in the MNF, and whose connectivity defines face geometry for ADAMS display.

ADDITIONAL OUTPUT

Additional Stress/Strain Plots Options (Case Control)

- **STRESS(PLOT)** Request stress shapes
- **GPSTRESS(PLOT)** Request grid point stress shapes to be included in the MNF file
- **STRAIN(PLOT)-** Request strain shapes
- **GPSTRAIN(PLOT)-** Request grid point strain shapes to be included in the MNF file

ADDITIONAL OUTPUT

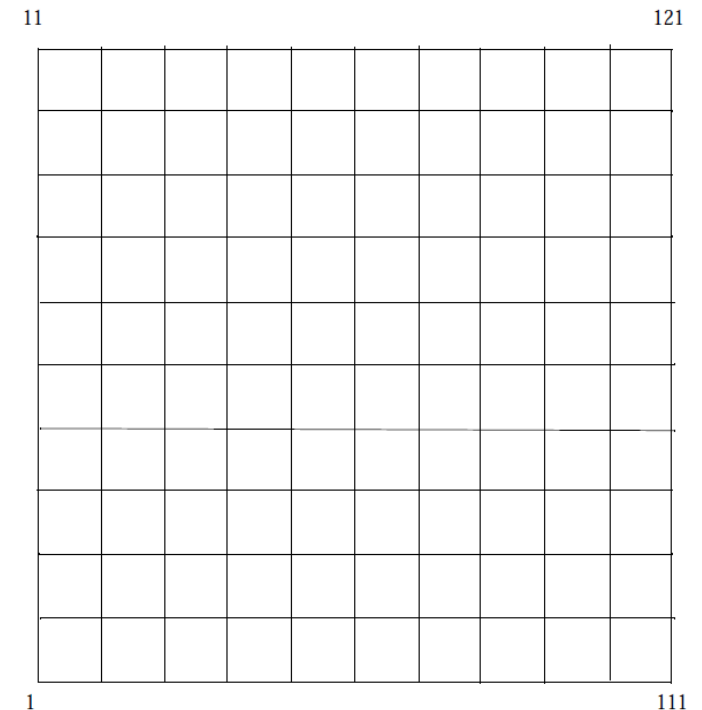
Additional Options (Case Control)

- OUTPUT(POST) Defines volume and surface for grid stress or strain shapes.
 - SET nnn=list - set of elements for surface definition for grid stress or strain shapes.
 - SURFACE nnn SET nnn NORMAL z3 - surface definition - Only one FIBER selection is allowed for all SURFACES.
 - VOLUME nnn SET nnn - volume definition.
 - The default SYSTEM BASIC is required with SURFACE or VOLUME.
- OUTPUT(PLOT) - necessary to define elements used to select grids to display the component in ADAMS.
 - SET nj = list of elements used to select grids to display the component.

EXAMPLE PROBLEM – SOL 103

SOL 103 Example

- **Create an Adams MNF file for the following model**
 - Attachment points at grid points 1,11,111, and 121 (all 6 DOFs)
 - Units (KG, N, M, SEC)
 - Request 10 modes
 - Define appropriate generalized coordinates
 - Define PLOTEs around the plate
 - Request stress and strain grid shapes
 - Request output2 file for durability analysis



SOL 103 - INPUT FILE

```
$ ADAMS/Durability with MSC.Fatigue requires op2 generation
$
ASSIGN OUTPUT2='e2_op2.out' STATUS=UNKNOWN UNIT=20 FORM=UNIFORM
$
SOL 103
CEND
$
TITLE= SIMPLE PLATE MODEL 10 X 10 ELEMENTS
$
$ Initiate an MSC.Nastran/ADAMS interface run
$ flexbody=yes is REQUIRED
$ psetid will define face geometry for adams component display
$ admout=yes also output op2 file for ADAMS/Durability with MSC.Fatigue
$ outstrs=yes output element stress shapes to op2 file
$
ADAMSMNF flexbody=yes, psetid=7772, admout=yes,
        outstrs=yes
$
$ Select real eigen value parameters - must be above or in first subcase
$
METHOD=300
$
$ Starting with v2003, residual vectors are by default always computed.
$ They should be controlled with the following case control command:
$ It is recommended that the MSC.Nastran defaults be taken.
$ The defaults result in numerically improved shapes.
$
RESVEC = COMPONENT
$
```

```
$
$ Generate stress and strain grid shapes
$
STRESS(PLOT) = ALL
STRAIN(PLOT) = ALL
GPSTRESS(PLOT) = ALL
GPSTRAIN(PLOT) = ALL
$
SUBCASE 5
LABEL= one atu overpressure - uniform
load=5
SUBCASE 50
LABEL= spin about edge (Y-Axis) at 1 revs/sec (60 RPM)
load =50
$
$ Set to define component shapes
$
OUTPUT(PLOT)
SET 7772 = 10001 THRU 10010
$
$ Define surface for stress and strain grid shapes
$
OUTPUT(POST)
SET 9998 = ALL
SURFACE 9998 SET 9998 NORMAL X3
$
```

SOL 103 - INPUT FILE (CONT.)

```
$
BEGIN BULK
$
$ ADAMS REQUIRES following DTI
$
DTI,UNITS,1,KG,N,M,SEC
$
$ Turn on gridpoint weight generator
$
PARAM,GRDPNT,0
$
$ Default value - ADAMS must use the above DTI,UNITS
$
PARAM,WTMASS,1.0
$
$ Select number of modes:
$
$
=====
EIGR      300      LAN                      10
$
=====
$
```

```
$
$ The corner grids 1, 11, 111, 121 are the exterior
$ or attachment point grids
$
ASET1,123456,1,11,111,121
$
$ SCALAR Point and QSET1 to define DOFs to use for component modes
$
SPOINT,80001,THRU,80018
QSET1,0,80001,THRU,80018
$
$ Define PLOTETs for set 7772 in output(plot) section
$
PLOTET,10001,1,12
PLOTET,10002,12,121
PLOTET,10003,121,111
PLOTET,10004,111,1
PLOTET,10006,2,10
PLOTET,10007,22,110
PLOTET,10008,120,112
PLOTET,10009,12,100
$
$ Get model data and load data
$
include 'modell.dat'
$
ENDDATA
```



ADAMSMNF

SOL 400

INTRODUCTION

- **The commands are similar to the those used in SOL 103**
- **Preloads are allowed by multiple steps input**
 - 1st STEP – preload
 - 2nd STEP – Modes calculations.
- **Some of these features are available using a SOL 106 to SOL 103 restart**
 - Not discussed here
 - See MSC Nastran Reference Manual if interested

INPUT RELATED TO MNF FILE CREATION IN SOL 400

- **Executive Control**

SOL 400

- **Typical Case Control**

AdamsMNF flexbody=yes

\$

SUBCASE 1

STEP 10

ANALYSIS = nlstatics

.

STEP 20

ANALYSIS = modes

METHOD = 100

RESVEC = component

Basic command needed for
MNF file (see QRG for
additional options)

- **Bulk Data Section**

- Bulk data entries for a SOL 400 run includes
 - Nonlinear static (BCONTACT, NLPARM, NLSTEP, etc.)
 - Modal perturbation (EIGRL) analysis
 - Sufficient SPOINTs to bring down component modes
 - Units for Adams

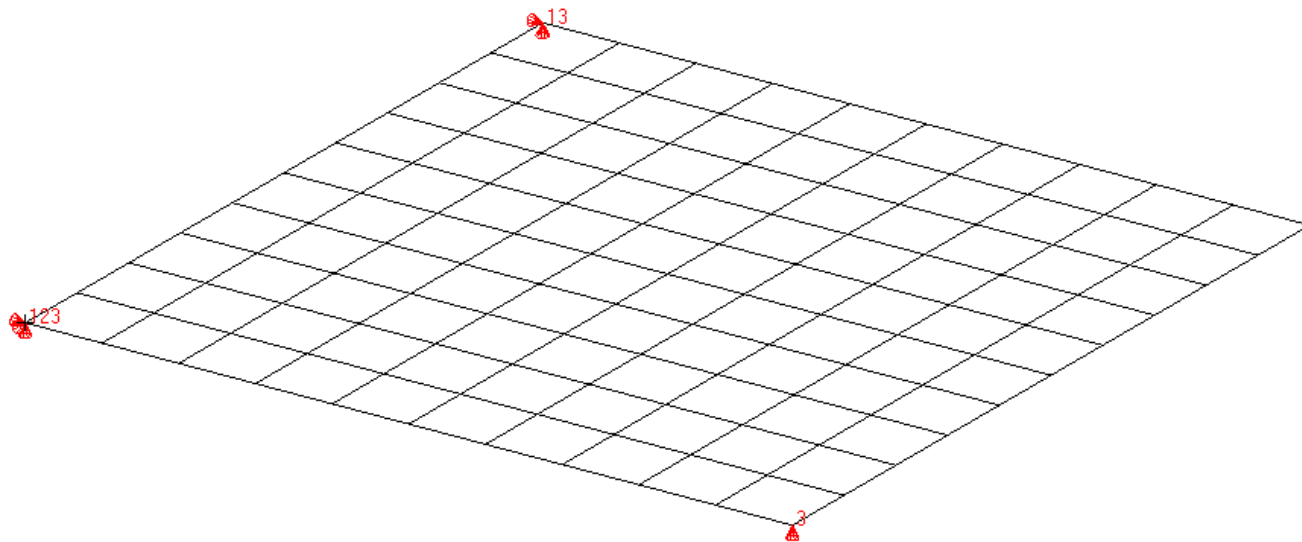
plus

Additional Bulk data section that contains the:

```
Begin Bulk Flxbdy = x
qseti,0,.....
aseti,123456,.....
```

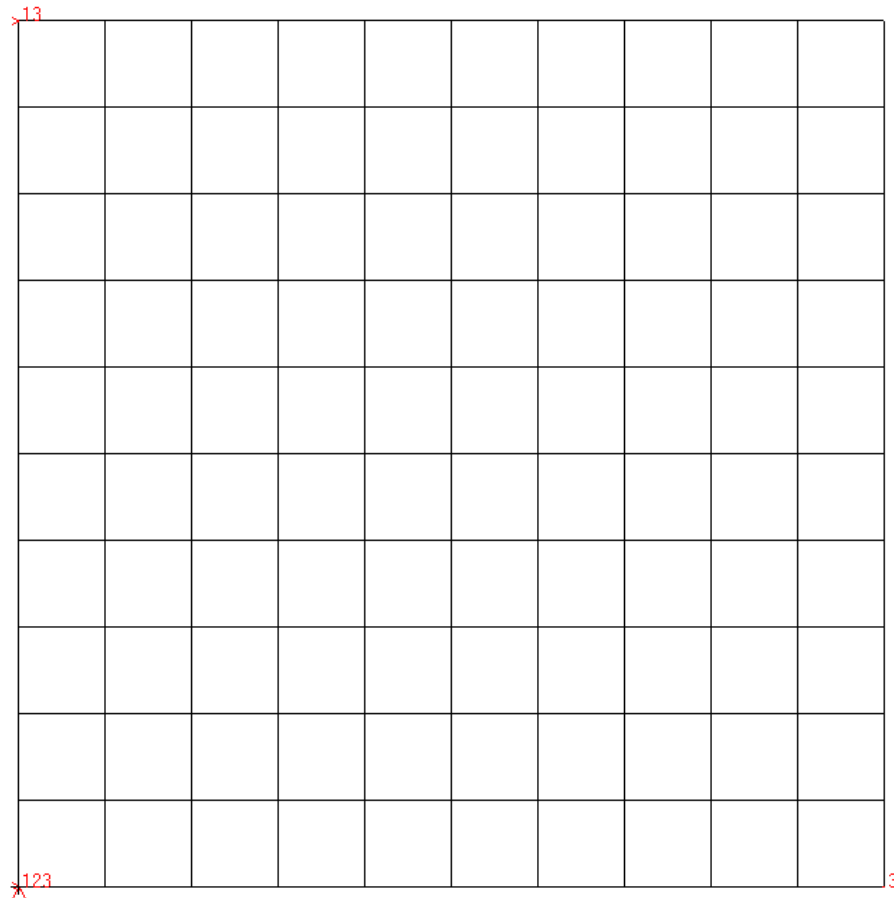
EXAMPLE PROBLEM

- **Plate model with preload**
 - Statically determinate set of constraints at the 3 corners
 - Preload with follower forces at the two ends



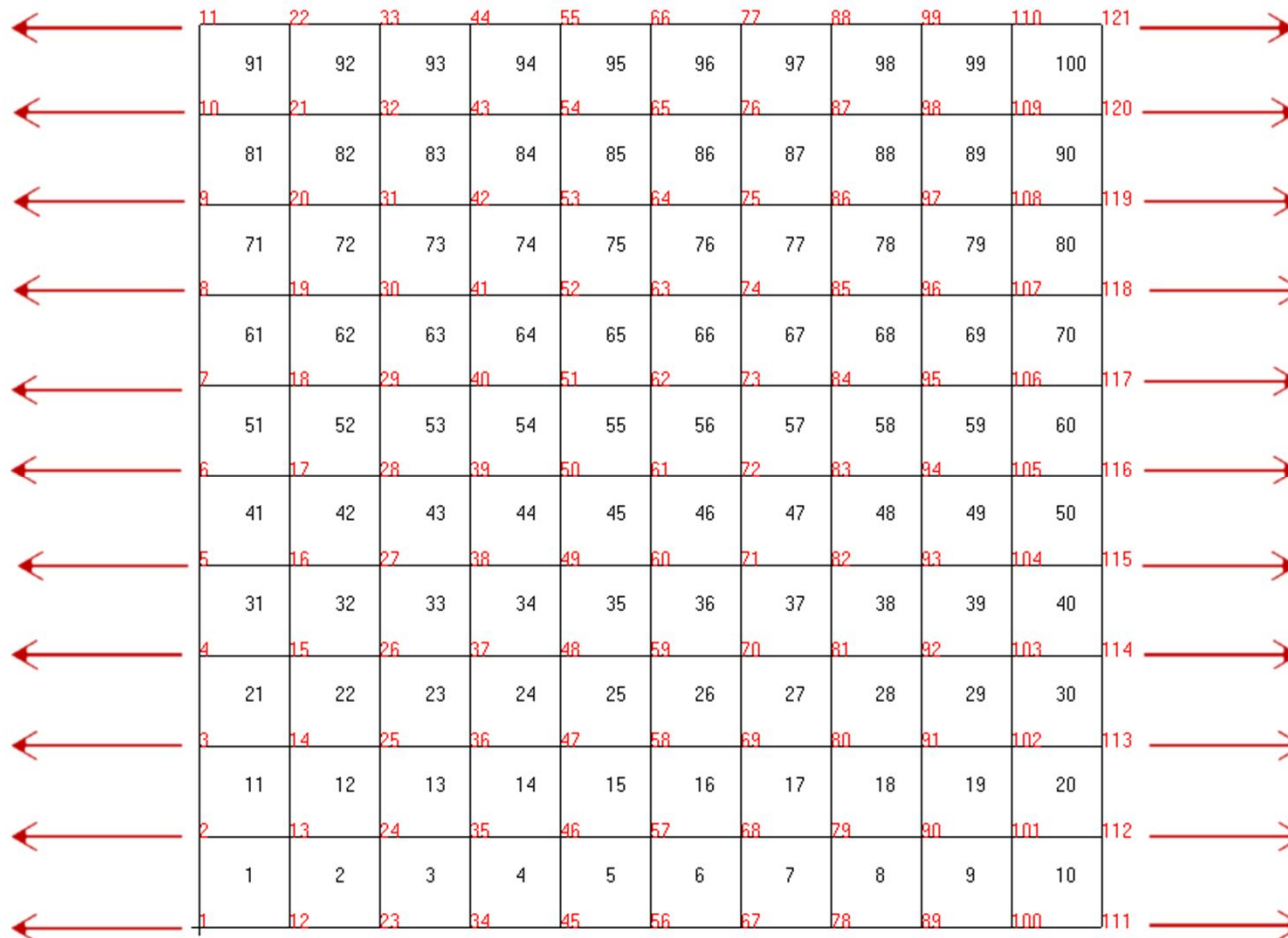
EXAMPLE PROBLEM (CONT.)

- **Plate Model**
 - Constraints



EXAMPLE PROBLEM (CONT.)

- **Plate Model**
 - Follower force (FORCE1 entries)



EXAMPLE PROBLEM (CONT.)

```

SOL 400
CEND
TITLE= SIMPLE PLATE MODEL 10 X 10 ELEMENTS SOL 400 NL PRELOAD
$$
$ Initiate an MSC.Nastran/ADAMS interface run
$ FLEXBODY=YES is REQUIRED
$ ADMOUT=YES also output op2 file
$ OUTGSTRS=YES output element stress shapes for Adams
$ OUTGSTRN=YES output element stress shapes for Adams
$
ADAMSMNF FLEXBODY=YES, ADMOUT=YES, OUTGSTRS=YES,
OUTGSTRN=YES, FLEXONLY=NO
$
SUBCASE 100
STEP 100
ANALYSIS=NLSTATICS
NLPARM = 1
SPCF = ALL
SPC = 100 $
LOAD=100
STRESS(PLOT) = ALL
STRAIN(PLOT) = ALL
GPSTRESS(PLOT) = ALL $
GPSTRAIN(PLOT) = ALL $
NLSTRESS(PLOT) = NONE $
$
STEP 200
ANALYSIS = modes
$
METHOD=300
$
$ Turn on residual vectors
$
RESVEC = COMPONENT
DISP(PLOT)=ALL
STRESS(PLOT) = ALL
STRAIN(PLOT) = ALL
GPSTRESS(PLOT) = ALL $
GPSTRAIN(PLOT) = ALL $
$

```

```

$
$ Define surface for stress and strain grid shapes
$
OUTPUT(POST)
SET 9998 = ALL
SURFACE 9998 SET 9998 FIBRE Z2 NORMAL X3
$
BEGIN BULK
$
PARAM,LGDISP,1
NLPARM,1,4,,,,,UPW,YES
$
PARAM,GRDPNT,0
$
$2345678 2345678 2345678 2345678 2345678 2345678 2345678 2345678
$ =====
EIGR      300      LAN                      10
$ =====
$
$ SCALAR Point to define DOFs to use for component modes
SPOINT,80001,THRU,80018
$
$ ADAMS REQUIRES following DTI
$
DTI,UNITS,1,KG,N,M,SEC
$
$ Add in plate tensioning follower load
$
$2345678 2345678 2345678 2345678 2345678 2345678 2345678 2345678
FORCE1 100      111      30000.  1      111
FORCE1 100      112      60000.  2      112
.
FORCE1 100      121      30000.  11     121
$
FORCE1 100      1      30000.  1      11 1

```

EXAMPLE PROBLEM (CONT.)

```
$
SPC1,100, 123, 1
SPC1,100, 13, 11
SPC1,100, 3, 111
$
$ PLUS REST OF THE BULK DATA
.
$
$ New Bulk Flxbdy bulk data section
$
BEGIN BULK FLXBDY = 77
$
$ The corner grids 1, 11, 111, 121 are the exterior
$ or attachment point grids
$
ASET1,123456,1,11,111,121
$ QSET1 to define DOFs to use for component modes
QSET1,0,80001,THRU,80018
$
ENDDATA
```

Attachment Points

Generalized Coordinates

EXAMPLE PROBLEM (CONT.)

- Modes Output**

SUBCASE 100 STEP 200						
MODE NO.	EXTRACTION ORDER	EIGENVALUE	R E A L E I G E N V A L U E S		GENERALIZED MASS	GENERALIZED STIFFNESS
			RADIANS	CYCLES		
1	1	-2.441036E+01	4.940684E+00	7.863342E-01	1.000000E+00	-2.441036E+01
2	2	-1.220921E+01	3.494169E+00	5.561142E-01	1.000000E+00	-1.220921E+01
3	3	-7.333120E-07	8.563364E-04	1.362902E-04	1.000000E+00	-7.333120E-07
4	4	-6.585049E-08	2.566135E-04	4.084131E-05	1.000000E+00	-6.585049E-08
5	5	1.441072E-08	1.200446E-04	1.910570E-05	1.000000E+00	1.441072E-08
6	6	2.361914E-07	4.859953E-04	7.734854E-05	1.000000E+00	2.361914E-07
7	7	4.393842E+04	2.096149E+02	3.336125E+01	1.000000E+00	4.393842E+04
8	8	1.189452E+05	3.448843E+02	5.489004E+01	1.000000E+00	1.189452E+05
9	9	2.876199E+05	5.363021E+02	8.535513E+01	1.000000E+00	2.876199E+05
10	10	4.641084E+05	6.812551E+02	1.084251E+02	1.000000E+00	4.641084E+05

GUIDELINES AND LIMITATIONS FOR SOL 400

- **SPOINTs MUST BE IN THE MAIN BULK DATA SECTION**
- **QSETi and ASETi must be in the BEGIN BULK FLXBDY section**
- **Can only have one BEGIN BULK FLXBDY section**
- **If there is contact in the model, bilinear Coulomb friction should be turned on**
 - **BCPARA,0,FTYPE,6**
- **If there are RTYPE elements in the model, RIGID=LAGRANGE (default) should be used**
- **If a DOF on the RTYPE element is on the ASET, make sure that it is the independent DOF.**
- **Stress and strain output are not supported for the advanced nonlinear elements**
- **Stresses and strains are incremental from the deformed state**

GUIDELINES AND LIMITATIONS FOR SOL 400 (CONT.)

- **AUTOQSETs are not supported**
- **Only a single Adams MNF is supported**
- **Transient analysis (NLTRAN) is not supported when using Adams MNF**
- **Should always create a free-free model for the modes**
- **Due to limitation NAS-13641, the rigid body may not be very clean**
 - Make sure that Adams de-activate the proper rigid body modes

